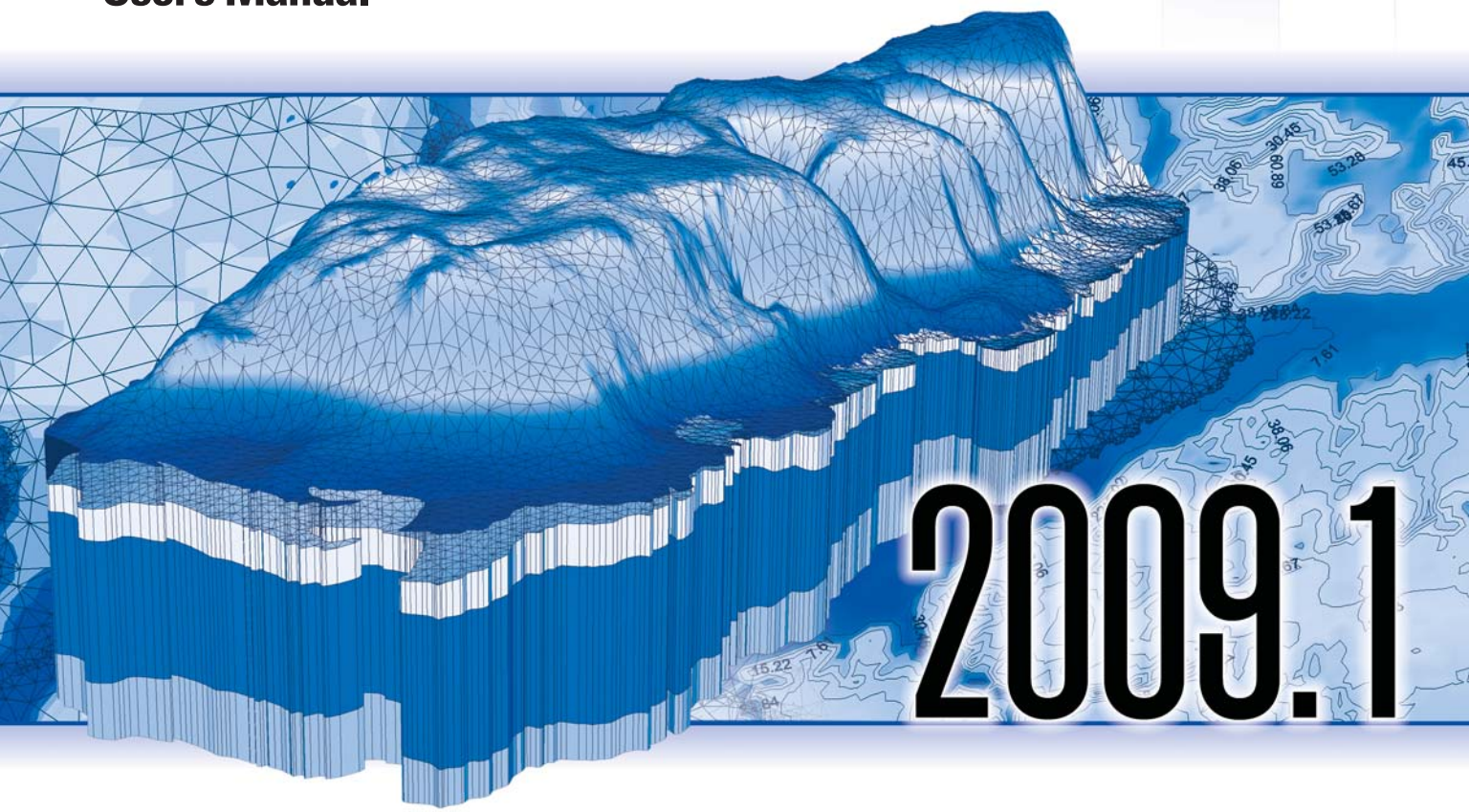
Schlumberger
WATER SERVICES

User's Manual



Hydro GeoBuilder

A flexible, simulator-independent, hydrogeological modeling environment

Copyright Information

© 2009 Schlumberger Water Services. All rights reserved.
No portion of the contents of this publication may be reproduced or transmitted in any form or by any means without the express written permission of Schlumberger Water Services.

Printed in Canada
2009

Table of Contents

1. Introduction to Hydro GeoBuilder 1

Hydro GeoBuilder Features 2

Hydro GeoBuilder Installation 5
- Hardware Requirements 5
- Installing Hydro GeoBuilder 6
- Uninstalling Hydro GeoBuilder 6
- Licensing 7
- Starting Hydro GeoBuilder 7

Hydro GeoBuilder Workflow 7

Hydro GeoBuilder User Interface 10
- Main Window 10
- Viewer Types 11
- Selecting Data Objects 12

Hydro GeoBuilder On-Line Help 12

2. Project Management 15

Creating a New Project 15

Modifying Project Settings 18

Saving a Project 21

Opening a Project 21

3. Importing Data 23

Importing Points 24

Importing Polylines 30

Importing Polygons 32

Importing Surfaces 33

Importing Wells 34

Importing 3D Gridded Data 41

Importing Cross Sections 44

Importing Maps ... 46
Georeferencing Images ... 48
Importing Time Schedules ... 53
Deleting Data Objects ... 56
Exporting Data Objects ... 56
Importing Data from Existing VMOD Models ... 57
4. Data Settings and Properties ... 61
Viewing General Data Settings ... 62
Viewing the Data Table ... 63
General ... 63
Well Table ... 65
Performing Operations on Data ... 70
Arithmetic Operations ... 71
Attribute Operations ... 72
Converting Model Layers to Points Data Object ... 74
Converting Well Tops to Points Data Object ... 74
Calculating Well Head Elevation (Z) from a Surface ... 76
Modifying Data Style Settings ... 76
Points, Polygons & Polylines ... 77
Cross Sections ... 83
Wells ... 85
Surfaces ... 88
3D Gridded Data ... 90
5. Data Viewers ... 95
Opening a New 2D or 3D Viewer ... 95
Working with Viewers ... 96
Displaying Data in Viewers ... 99
Modifying Viewer Settings ... 100
Viewer Controls ... 104
Linking 2D Viewers with Attribute Tables ... 105
Exporting Viewers ... 105

Creating a New Data Object 106
Digitizing & Editing Geometry in 2D Viewers 107
6. Creating Surfaces 109
Creating a New Surface 110
Surface Settings 114
Deleting a Surface 114
7. Creating a Conceptual Model 117
Creating a New Conceptual Model 118
Conceptual Model Settings 119
Deleting a Conceptual Model 120
8. Defining Horizons 121
Creating Horizons 124
Editing and Deleting Horizons 126
Viewing Structural Zones 127
9. Property Modeling 129
Defining a New Property Zone 130
Editing Property Zones 135
Property Zone Settings 136
Deleting a Property Zone 136
10. Simulation Model Domain 139
Creating the Simulation Model Domain 139
11. Boundary Modeling 141
Boundary Conditions Overview 142
Pumping Well 142
Specified Head 143
River 144
General Head 146
Drain 149

Recharge 151
Evapotranspiration 151
Lake 152
Specified Flux 154

Defining a New Boundary Condition 155
Defining a Pumping Well Boundary Condition 155
Defining Other Boundary Conditions 158

Modifying Boundary Conditions 172

Deleting Boundary Conditions 173

12. Model Domain Discretization 175

Creating a Finite Difference Grid 175
Defining the Horizontal Grid 176
Defining the Vertical Grid 177

Editing a Finite Difference Grid 182
Horizontal Grid Refinement 183

Deleting a Numerical Grid 184

Creating a Finite Element Mesh 184
Defining the Horizontal Mesh Settings 186
Defining Slice Elevations 190

13. Translating to Numerical Model 193

Translating a Conceptual Model to a Numerical Model 193
Translating to MODFLOW 193
Translating to FEFLOW 198

Importing into Visual MODFLOW 203

Running MODFLOW-LGR Simulation 203

Viewing Results in Hydro GeoBuilder 205

14. Appendix A: Supported Data Types 207

15. Index 209

Preface

Schlumberger Water Services (SWS) is a recognized leader in the development and application of innovative groundwater technologies in addition to offering expert services and professional training to meet the advancing technological requirements of today's groundwater and environmental professionals.

Waterloo Hydrogeologic Software (WHS) consists of a complete suite of environmental software applications engineered for data management and analysis, modeling and simulation, visualization, and reporting. WHS is currently developed by SWS and sold globally as a suite of desktop solutions.

For over 18 years, our products and services have been used by firms, regulatory agencies, and educational institutions around the world. We develop each product to maximize productivity and minimize the complexities associated with groundwater and environmental projects. To date, we have over 14,000 registered software installations in more than 85 countries!

Need more information?

If you would like to contact us with comments or suggestions, you can reach us at:

Schlumberger Water Services
460 Phillip Street - Suite 101
Waterloo, Ontario, CANADA, N2L 5J2

Phone: +1 (519) 746-1798
Fax: +1 (519) 885-5262

General Inquiries: sws-info@slb.com

Web: www.swstechnology.com, www.water.slb.com

Obtaining Technical Support

To help us handle your technical support questions as quickly as possible, please have the following information ready before you call, or include it in a detailed technical support e-mail:

- A complete description of the problem including a summary of key strokes and program event (or a screen capture showing the error message, where applicable)
- Product name and version number
- Product serial number
- Computer make and model number
- Operating system and version number
- Total free RAM
- Number of free bytes on your hard disk

- Software installation directory
- Directory location for your current project files

You may send us your questions via e-mail, fax, or call one of our technical support specialists. Please allow up to two business days for a response. Technical support is available 8:00 am to 5:00 pm EST Monday to Friday (excluding Canadian holidays).

Phone: +1 (519) 746-1798
Fax: +1 (519) 885-5262

E-mail: sws-support@slb.com

Training and Consulting Services

Schlumberger Water Services offers numerous, high quality training courses globally. Our courses are designed to provide a rapid introduction to essential knowledge and skills, and create a basis for further professional development and real-world practice. Open enrollment courses are offered worldwide each year. For the current schedule of courses, visit: www.swstechnology.com/training or e-mail us at: sws-training@slb.com.

Schlumberger Water Services also offers expert consulting and peer reviewing services for data management, groundwater modeling, aqueous geochemical analysis, and pumping test analysis. For further information, please contact sws-services@slb.com.

Waterloo Hydrogeologic Software

We also develop and distribute a number of other useful software products for the groundwater professionals, all designed to increase your efficiency and enhance your technical capability, including:

- Visual MODFLOW Premium*
- Hydro GeoAnalyst*
- Aquifer Test Pro*
- AquaChem*
- GW Contour*
- UnSat Suite Plus*
- Visual HELP*
- Visual PEST-ASP

Visual MODFLOW Premium

Visual MODFLOW Premium is a three-dimensional groundwater flow and contaminant transport modeling application that integrates MODFLOW-2000, SEAWAT-2000, MODPATH, MT3DMS, MT3D99, RT3D, VMOD 3D-Explorer, WinPEST, Stream Routing Package, Zone Budget, MGO,

SAMG, and PHT3D. Applications include well head capture zone delineation, pumping well optimization, aquifer storage and recovery, groundwater remediation design, simulating natural attenuation, and saltwater intrusion.

Hydro GeoAnalyst

Hydro GeoAnalyst is an information management system for managing groundwater and environmental data. Hydro GeoAnalyst combines numerous pre and post processing components into a single program. Components include, Project Wizard, Universal Data Transfer System, Template Manager, Materials Specification Editor, Query Builder, QA/QC Reporter, Map Manager, Cross-Section Editor, HGA 3D-Explorer, Borehole Log Plotter, and Report Editor. The seamless integration of these tools provide the means for compiling and normalizing field data, analyzing and reporting subsurface data, mapping and assessing spatial information, and reporting site data.

AquiferTest Pro

AquiferTest Pro, designed for graphical analysis and reporting of pumping test and slug test data, offers the tools necessary to calculate an aquifer's hydraulic properties such as hydraulic conductivity, transmissivity, and storativity. AquiferTest Pro is versatile enough to consider confined aquifers, unconfined aquifers, leaky aquifers, and fractured rock aquifers conditions. Analysis results are displayed in report format, or may be exported into graphical formats for use in presentations. AquiferTest Pro also provides the tools for trends corrections, and graphical contouring water table drawdown around the pumping well.

AquaChem

AquaChem is designed for the management, analysis, and reporting of water quality data. AquaChem’s analysis capabilities cover a wide range of functions and calculations frequently used for analyzing, interpreting and comparing water quality data. AquaChem includes a comprehensive selection of commonly used plotting techniques to represent the chemical characteristics of aqueous geochemical and water quality data, as well includes PHREEQC - a powerful geochemical reaction model.

GW Contour

The GW Contour data interpolation and contouring program incorporates techniques for mapping velocity vectors and particle tracks. GW Contour incorporates the most commonly used 2D data interpolation techniques for the groundwater and environmental industry including Natural Neighbor, Inverse Distance, Kriging, and Bilinear. GW Contour is designed for contouring surface or water levels, contaminant concentrations, or other spatial data.

UnSat Suite Plus

UnSat Suite Plus seamlessly integrates multiple one-dimensional unsaturated zone flow and solute transport models into a single, intuitive working environment. Models include SESOIL, VS2DT,

VLEACH, PESTAN, Visual HELP and the International Weather Generator. The combination of models offers users the ability for simulating the downward vertical flow of water and the migration of dissolved contaminants through the vadose zone. UnSat Suite Plus includes tools for project management, generating synthetic weather data, modeling flow and contaminants through the unsaturated zone, estimating groundwater recharge and contaminant loading rates, and preparing compliance reports.

Visual HELP

Visual HELP is a one-dimensional, unsaturated zone flow modeling application built for optimizing the hydrologic design of municipal landfills. Visual HELP is based on the US E.P.A . HELP model (Hydrologic Evaluation of Landfill Performance) and has been integrated into a 32-Bit Windows application. It combines the International Weather Generator, Landfill Profile Designer, and Report Editor. Applications include designing landfill profiles, predicting leachate mounding, and evaluating potential leachate seepage to the groundwater.

Visual PEST-ASP

Visual PEST-ASP combines the powerful parameter estimation capabilities of PEST-ASP, with the graphical processing and display features of WinPEST. Visual PEST-ASP can be used to assist in data interpretation, model calibration and predictive analysis by optimizing model parameters to fit a set of observations. This popular estimation package achieves model independence through its capacity to communicate with a model through its input and output files.

Groundwater Instrumentation

Diver-NETZ

Diver-NETZ is an all-inclusive groundwater monitoring network system that integrates high-quality field instrumentation with the industries latest communications and data management technologies. All of the Diver-NETZ components are designed to optimize your project workflow from collecting and recording groundwater data in the field - to project delivery in the office.

*Mark of Schlumberger

1

Introduction to Hydro GeoBuilder

Hydro GeoBuilder is a powerful software package that provides the tools for building three-dimensional groundwater conceptual models using raw GIS data objects. The conceptual model approach to groundwater modeling allows you to:

- **Build a conceptual model of the groundwater system, prior to the simulation** - The geological formations, property model, and boundary conditions are all designed outside the model grid or mesh; this allows the flexibility to adjust your interpretation of the groundwater system before applying a discretization method and converting to a numerical model.
- **Build the model with minimal data pre-processing required** - Working with grid-independent data allows you to maximize the use of your existing GIS data and incorporate physical geology and geographic conditions before designing a grid or mesh.
- **Generate and simulate regional and local-scaled models** - With support for MODFLOW-LGR package, you can design local grids around areas of interest, directly within the conceptual model environment. Calculated heads from a regional model can also be used as boundary conditions for local-scaled models.
- **Design the correct model faster** - The grid-independent raw data is left intact and is not constricted by grid cells or mesh elements when modifying the data and project objective. This allows you to generate multiple numerical models from the same conceptual model.
- **Make changes to the model data and immediately see results** - The conceptual model environment provides simultaneous 2D and 3D views which are updated whenever changes to the data are made.

Please see “Hydro GeoBuilder Features” on page 2 for a comprehensive list of features available in Hydro GeoBuilder.

This document provides detailed descriptions of all features and functionality available in Hydro GeoBuilder.

1.1 Hydro GeoBuilder Features

General Features

- Supports the following coordinate systems:
  - Geographic coordinate systems (data import only)
  - Projected coordinate systems
  - Local Cartesian

Work With Grid-Independent Data

- Import spatial and attribute data from a wide variety of data types including:
  - Points (.XLS, .TXT, .CSV, .MDB, .SHP, .DXF, .TRP)
  - Polygons (.SHP, .DXF)
  - Polylines (.SHP, .DXF)
  - 3D Gridded Data (.HDS, .DAT)
  - Raster Images (.BMP, .TIF, .JPG)
  - Time Schedules (.XLS)
  - Surfaces (.DEM, .GRD, .TXT. ,.ASC)
  - Hydro GeoAnalyst (HGA) Cross Sections (.3XS)
  - Vertical and Horizontal Wells (.XLS)

- View and modify settings for imported data
  - View data object meta data including the source file name, field mappings and the native coordinate system
  - View raw attribute data in a spreadsheet view
  - Apply mathematical operations to data, e.g., set an attribute as a constant value, convert well tops to a points data object, and convert HGA cross section model layers to a points data object
  - Drape a raster image over a surface data object, e.g., digital elevation models
  - Set symbol properties for points, polygons, polylines and display labels using a variety of style options
  - Color render shape features by attribute value using a classified or stretched color scheme
  - Show contour lines and set color rendering options for surface layers
  - Add, remove and modify wells and associated well data including screens intervals, diver observation points, well tops, well paths (for horizontal only), and pumping schedules

- Create surfaces from points data objects
  - Using one or more points data objects, generate surface layers using

Inverse Distance, Kriging or Natural Neighbor interpolation methods
    - Configure the interpolation method by modifying various interpolation settings
    - Clip the generated surface to the horizontal extents of a specified polygon data object

- Digitize new data objects using 2D Viewer
    - Using the 2D Viewer editing tools, digitize a new polyline, polygon or points data object

2D & 3D Visualization

- Visualize data objects and conceptual model features using interactive 2D and 3D Viewers
    - Use various screen configurations to display multiple 3D or 2D Viewers simultaneously,e.g., cascade, tile horizontally/vertically
    - Zoom, rotate and move data within the viewer using your mouse
    - Modify viewer settings including the background color and vertical exaggeration (3D Viewer only)
    - In 3D Viewers, remove parts of the displayed data by creating cutaways along the X, Y and Z axis
    - In 2D Viewers, select individual data object features (points, line, shapes), and then view the corresponding attribute data in spreadsheet view, and vice versa
- Edit data object geometry in 2D Viewer
    - Modify existing data objects by manually digitizing points, polylines and polygons
    - Rotate, scale and delete shapes
    - "Undo" all edits and revert back to original shape

Define Multiple Conceptual Models

- Create multiple conceptual models with different interpretations, or copy existing conceptual models
- Define conceptual model geometry using imported data objects
    - Define the horizontal model boundary using an imported or digitized polygon data object
    - Create vertical horizons from surfaces that are either imported or created by interpolating raw XYZ points
    - Select from different horizon types to accommodate various geological conditions (pinchouts, discontinuous layers, etc.)
    - Automatically create 3D structural zones from defined horizons

Property Modeling

- Create property zones from imported or digitized polygon data objects, or from from generated structural zones
- Assign property values for conductivity, storage and initial heads using various methods:
    - Use a constant value
    - Map to imported polygon shapefile attributes
    - Map to imported 3D Gridded data attributes
    - Use surface data object

Boundary Modeling

- Automatically generate the simulation domain using the boundaries defined for the conceptual model
- Apply boundary conditions to the top, bottom, sides or an intermediate layer of the simulation model domain
- Support for the following boundary conditions:
    - Pumping Wells
    - Specified Head
    - River
    - General Head
    - Drain
    - Recharge
    - Evapotranspiration
    - Lake
    - Specified Flux
- For linear boundary conditions, define local zones from line segments using an interactive 2D Viewer window
- For linear boundary conditions, define parameters at start, end or intermediate vertices along a line, and interpolate values between each vertex
- Set each boundary condition parameter as static or transient
- Define boundary condition parameters using one or more of the following methods:
    - Use a constant value
    - Map to imported shapefile attributes
    - Use a surface data object
    - Use a time schedule data object (for transient boundary conditions)
    - Use attributes from 3D Gridded data objects

Model Discretization

- Discretize your model using the finite difference method or the finite element method.
- When working with finite difference grids:

- Specify the number of rows and columns, grid origin, and the angle of rotation
- Choose from the following finite difference grid types:
  - •Deformed
  - •Uniform
  - •Deformed-Uniform
- Perform horizontal grid refinement/coarsening within a user-defined row/column interval
- Define a child grid within a numerical grid for running Local Grid Refinement (LGR) simulations using the MODFLOW-LGR package

- When working with finite elements meshes:
  - Use imported shape data objects to define the superelement mesh
  - Choose from various Delaunay triangulation methods including constrained and conforming
  - Refine areas of the mesh using digitized or imported polygon shapes
  - Fit the mesh to your model domain using deformed or semi-uniform vertical slices

Numerical Modeling

- Once the simulation model domain has been defined, translate the conceptual objects to the simulation model grid, and create the necessary MODFLOW input files
- Support for MODFLOW-2000, MODFLOW-2005 and MODFLOW-LGR packages
- Translate property input using MODFLOW BCF or LPF packages
- Select which boundary condition packages to translate
- Import the generated MODFLOW files into Visual MODFLOW and run the simulation using MODFLOW-2000 or MODFLOW-2005. For Local Grid Refinement simulations, use the provided MODFLOW-LGR executable to run the model.
- Import the simulation results back into Hydro GeoBuilder for visualizing heads, path lines, contour lines, etc.

1.2 Hydro GeoBuilder Installation

1.2.1 Hardware Requirements

Hydro GeoBuilder requires the following minimum system configuration:

- Pentium 4+ 600MHz (1GHz recommended)
- 512 MB RAM (1GB or more recommended)
- CD ROM drive
- 100 MB of free hard drive space

- Graphics card with 3D Graphics Accelerator
- Windows XP Pro (SP3) 32-Bit; Windows XP Pro (SP2) 64-Bit; Windows Vista Business , Ultimate or Enterprise, 32-Bit (SP1) and 64-Bit. **Note**: Windows XP Home, Windows Vista Home Premium, Home Basic or Starter Versions, are not supported.
- Microsoft .NET Framework v.3.0 installed (provided with installation)

Note: If you intend to build complex projects, it is recommended that you upgrade to the recommended specifications in the above list.

If you have any problems with your particular system configuration, please contact your system administrator, or contact SWS technical support.

1.2.2 Installing Hydro GeoBuilder

Hydro GeoBuilder is distributed on one CD-ROM. To install, please follow these directions:

Note: For detailed installation instructions, please refer to the Hydro GeoBuilder Getting Started Guide.

- Place the CD into your CD-ROM drive and the initial installation screen should load automatically. Once loaded, an installation interface will be presented.
- On the installation screen, you may choose from the following two buttons: **Hydro GeoBuilder Installation** and **Hydro GeoBuilder User's Manual**
- The User's Manual button will display a PDF document of the manual, which requires the Adobe Reader to view. If you do not have the Adobe Reader, a link has been created in the interface to download the appropriate software.
- The Installation button will initiate the installation of Hydro GeoBuilder on your computer. Hydro GeoBuilder must be installed on your local hard disk in order to run. Follow the installation instructions, and read the on-screen directions carefully. You will be prompted to enter your name, company name and serial number. Please ensure that you enter your serial number exactly as it appears on your CD case or invoice. Be sure to use capital letters and hyphens in the correct locations.
- Once the installation is complete, you should see the Hydro GeoBuilder icon on your Desktop labeled VMOD Hydro GeoBuilder. To start working with Hydro GeoBuilder, double-click on this icon.

1.2.3 Uninstalling Hydro GeoBuilder

To uninstall Hydro GeoBuilder, follow the steps below:

- Make sure that Hydro GeoBuilder program is closed
- For Windows XP users, go to **Start / Settings / Control Panel**. For Windows Vista users, go to **Start / Control Panel**.

- Click **Add or Remove Programs** (for Windows XP) or **Programs** (for Vista)
- Select **Hydro GeoBuilder** from the list of installed programs.
- Click **Uninstall**.

1.2.4 Licensing

Hydro GeoBuilder supports both dongle-based hardware licensing and software-based licensing.

For more information on software licensing, please consult the **Hydro GeoBuilder Getting Started Guide**, available on the installation CD in PDF format.

For general license inquires, please contact Schlumberger Water Services Sales: **sws-sales@slb.com**

1.2.5 Starting Hydro GeoBuilder

Once Hydro GeoBuilder has been installed on your computer, simply double-click on the Hydro GeoBuilder shortcut icon (shown below), located on your computer's desktop.

Alternatively, you can access the software via the start menu by clicking on **Start/ Programs/SWS Software/Visual MODFLOW/Hydro GeoBuilder.**

Note: If you are using dongle-based hardware licensing, please ensure that your dongle is connected to your computer (AFTER you have installed the software), and that you have properly configured your installation.

1.3 Hydro GeoBuilder Workflow

In general, the Hydro GeoBuilder workflow is comprised of the following tasks: import raw data, create a conceptual model, generate the simulation model domain, define the numerical model, and then run the simulation and analyze results. Each task is described below.

Import Raw Data

Start by importing raw geographic and hydrogeologic field data into your project using the flexible Hydro GeoBuilder import utility. Some examples of data that can be imported include:

- **Vertical or Horizontal Well**s - including well heads, screen intervals, pumping schedules, observation points, well tops, observation data and well path information.

- **Surfaces -** *e.g.*, digital elevation models
- **Shapes** - including polygons, polylines and points.
- **Cross Sections**- geology, hydrogeology and model interpretation data generated in Hydro GeoAnalyst software.
- **3D Gridded data -** including TecPlot grids, or MODFLOW heads (.HDS).
- **Site maps** - raster images, *e.g*., satellite imagery, topographic maps or aerial photographs of model region.
- **Time Schedules** - *e.g.*, pumping schedules, hydrographs

Once the data is imported you can manipulate the geometry and attribute data by applying arithmetic or geometric operations, or by manually editing data using the interactive 2D Viewer.

Related chapters:

- Chapter 2: Project Management
- Chapter 3: Importing Data
- Chapter 4: Data Settings and Properties
- Chapter 5: Data Viewers
- Chapter 6: Creating Surfaces

Create a Conceptual Model

Once you have imported your data into Hydro GeoBuilder you can create your conceptual model using the data as building blocks. Define the geometry of the conceptual model using a polygon and surface data objects. Create property zones using structural zones and polygons, and assign property parameter attributes using constant values or attribute data from surfaces, 3D gridded data and shapefiles.

Related chapters:

- Chapter 7: Creating a Conceptual Model
- Chapter 8: Defining Horizons
- Chapter 9: Property Modeling

Generate the Simulation Domain

Once you have created the conceptual model, generate the simulation model domain, which represents the full vertical and horizontal extents of the model area. Once this is generated, you can assign boundary conditions to the top, bottom or sides using imported or digitized polygon/polyline data objects, and then assign boundary condition attributes using constant values or attribute data from imported surfaces, 3D grids and time-schedules.

Related chapters:

- Chapter 10: Simulation Model Domain
- Chapter 11: Boundary Modeling

Define the Numerical Model

Choose to discretize your conceptual model using the finite difference method (grid) or the finite element method (mesh), and then translate the conceptual model information to a numerical model using the translation wizard. Hydro GeoBuilder automatically generates the appropriate MODFLOW or FEFLOW input files and saves them to the data repository.

Related chapters:

- Chapter 12: Creating a Finite Difference Grid
- Chapter 13: Translating to Numerical Model

Run the Simulation & Analyze Results

Import your model into the simulator of your choice. If you chose to create a finite difference model, import the translated MODFLOW files into Visual MODFLOW and run the simulation using one of the numeric engines. Model output results can be analyzed in Visual MODFLOW, or imported back into Hydro GeoBuilder for visualizing in a 3D Viewer. If you chose to create a finite element model, import the translated .FEM file into FEFLOW to make further changes, or run the simulation.

Related chapters:

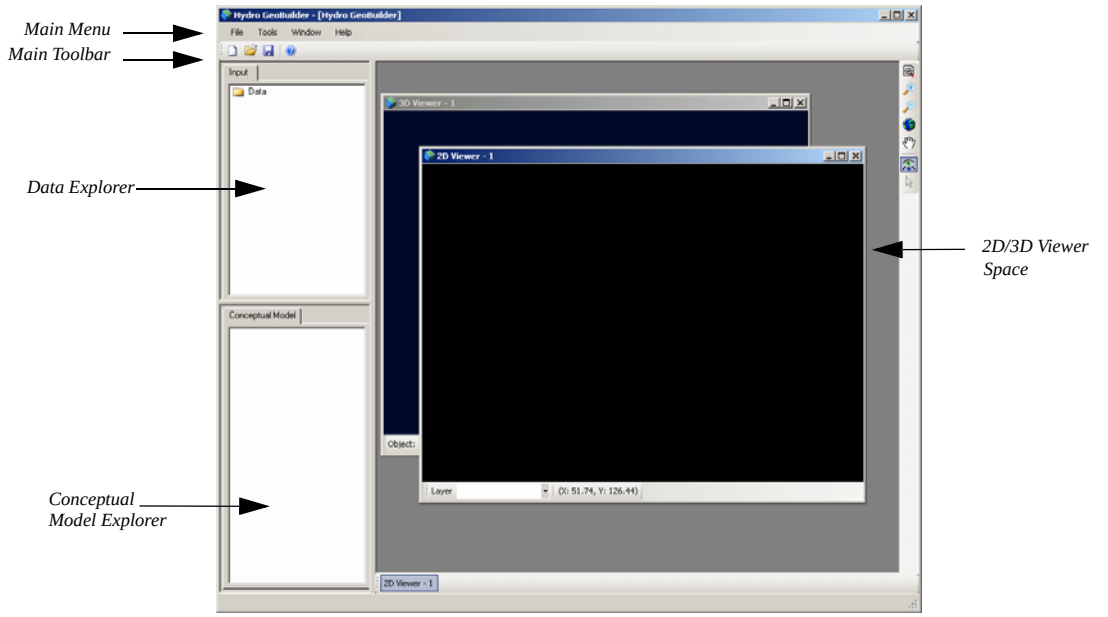
- Chapter 13: Importing into Visual MODFLOW

Note: **Hydro GeoBuilder is fully supported by Visual MODFLOW 4.3**. Using with older versions is not recommended, however you may attempt to use Hydro GeoBuilder with Visual MODFLOW 4.0 and higher.

1.4 Hydro GeoBuilder User Interface

1.4.1 Main Window

The Hydro GeoBuilder main window is shown below, followed by a descriptions of the different sections.



Main Menu: Provides access to various menu commands, *e.g.*, Save, Open, New, New 3D Window, New 2D Window, Help etc.

Main Toolbar: Provides short-cut buttons for various commands, *e.g.*, **New** project, **Open** project, **Save Project** and view **Online Help**.

Data Explorer: Allows you to manage imported and digitized data objects. When a project is opened, right-clicking in this space will display a pop-up menu, which allows you to perform various tasks, including:

- Import data
- View spreadsheet table
- Create new data objects
- Modify data object settings and properties
- Display data in 3D or 2D Viewers
- Export data to .SHP or .CSV file
- Create surfaces from points
- Create folders

Conceptual Model Explorer: When a new conceptual model is created, this space contains the conceptual model tree. From the conceptual model tree you can perform various tasks, including:

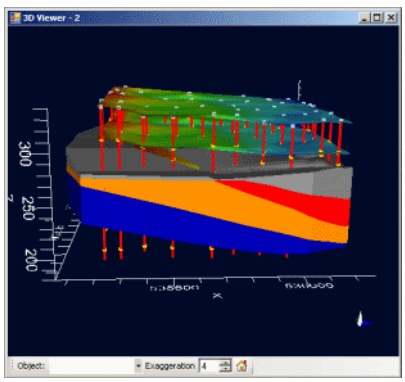
- Create horizons and structural zones
- Property modeling
- Create the simulation domain
- Define various boundary conditions
- Create numerical models
- Translate conceptual model to a numerical model
- View conceptual model data in 2D and 3D Viewers

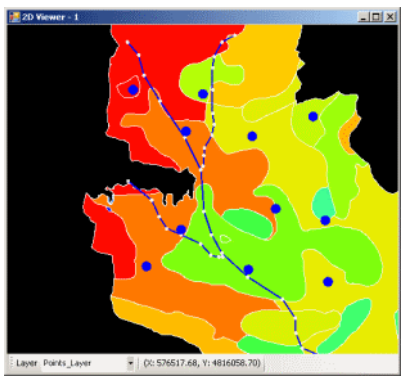
2D/3D Viewer Space: This space contains all opened 2D or 3D Viewers.

1.4.2 Viewer Types

Hydro GeoBuilder supports two types of interactive viewers: 3D Viewer and 2D Viewer. The **3D Viewer** is based on OpenGL technology, allowing you to visualize graphically-rich representations of your groundwater model from an oblique perspective. The **2D Viewer** allows you to view your data from a planar perspective, and provides various tools for editing and drawing data objects.

Hydro GeoBuilder allows you to have multiple viewers opened and displayed simultaneously. Both viewers can be launched by clicking on **Window** from the main menu, and then selecting **New 2D Window** or **New 3D Window**.

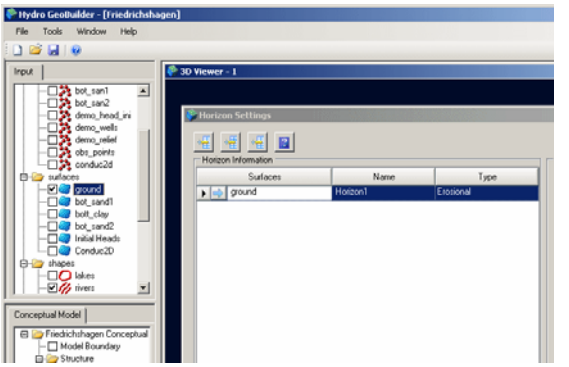




For more information on Hydro GeoBuilder viewers, please see “Data Viewers” on page 95.

1.4.3 Selecting Data Objects

Many wizards and dialog boxes in Hydro GeoBuilder require you to select data objects from the **Data Explorer** or **Conceptual Model Explorer**, *e.g.*, when defining horizons, creating property zones, and assigning attributes to boundary conditions. When you see a **Blue Arrow** located next to an input field in a dialog box or a wizard, this means that a data object selection is required. Simply click the appropriate data object from the **Data Explorer** or **Conceptual Model Explorer**, and then click the **Blue Arrow** button to insert the data object into the input field.



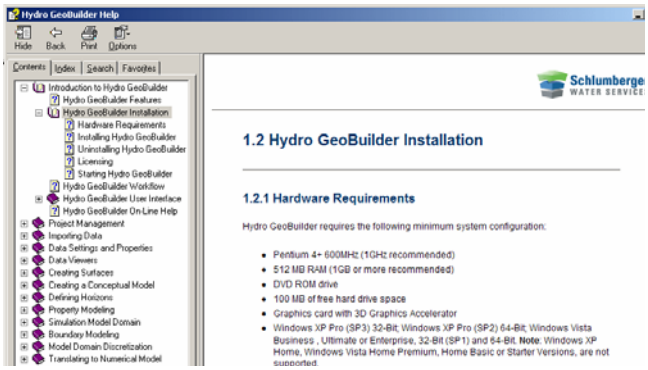
1.5 Hydro GeoBuilder On-Line Help

The Hydro GeoBuilder online help includes full color screen captures and illustrations which are black and white in this User's Manual . The information of interest can be viewed at all stages of the modeling project.

The online help can be access by selecting **Help/Help Topics** from the top menu bar, or by clicking the button from the top tool bar. Moreover, most dialog boxes and windows contain a **[Help]** button that when clicked, will open the relevant section of the online help.

The On-line help window (shown below) is divided into three main areas:

- A **Navigation Frame** on the left display the Contents, Index, Search, and Favorites tabs.
- A **Toolbar** across the top displays a set of buttons to help navigate through the online help.
- A **Topic Frame** on the right displays the actual Help topics included in the Online Help.



The tabs in the Navigation Frame provide the core navigational features as described below.

Contents

The **Contents** tab displays the headings in the “Table of Contents” in the form of an expandable/collapsible tree. Closed book icons represent Table of Contents headings that have sub-headings.

Index

The **Index** tab displays the list of Help topics. You can scroll to find the index entry you want, or you can type in the first few letters of the keyword in the text box, and the index will scroll automatically as you type. Double-click an index entry to display the corresponding Help topic. Alternatively, you may select an index entry and then click Display button to open the Help topic.

Search

The **Search** tab is used to search the On-Line Help documents for a word or phrase of interest. Simply type the search word(s) or phrase(s), then press **<Enter>** or click the **Display** button.

Favorites

You can add frequently accessed Help topics to a personal list of favorites, which is displayed in the **Favorites** tab. Once you have added a topic to your list of favorites, you can access the topic by double-clicking it. Click **Add** to add the currently displayed topic to your favorites list. Select a favorite and then click **Remove** to delete a topic from your favorites list.

2

Project Management

In Hydro GeoBuilder, a project refers to one or more conceptual models that share a common general feature, *e.g*., the same area, or the same simulation objective, etc.

When creating a new project, you may specify various project settings such as the data repository location, the project's coordinate system/datum and the default units. This process is described in the following sections.

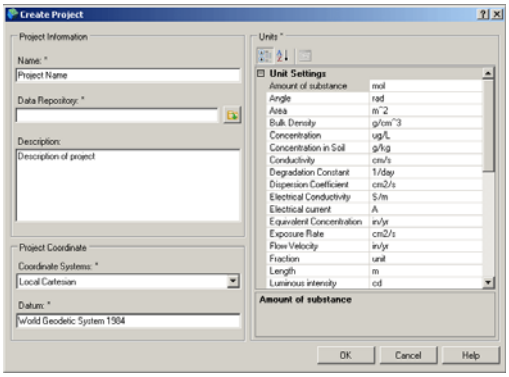
This chapter presents information on the following topics:

- Creating a New Project
- Modifying Project Settings
- Saving a Project
- Opening a Project

2.1 Creating a New Project

To create a new project, follow the steps below

- From the main menu, go to **File** > **New** > **Project...** or select the **New** button from the toolbar.
- Specify the project settings in the **Create Project** dialog. Each setting is described in the following sections.



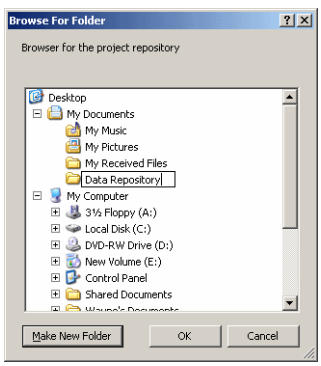
Project Information

Name

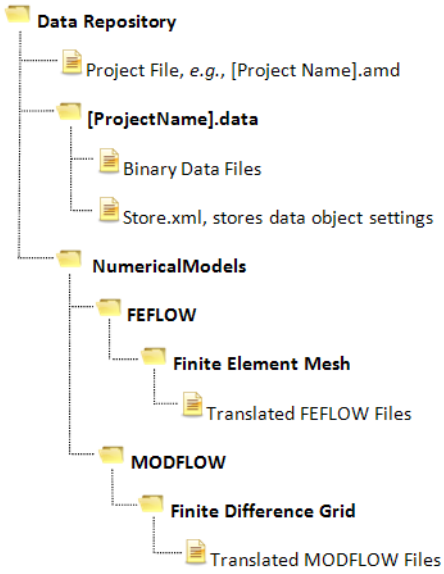
Type a unique name for your project. The project name cannot contain the characters \ / : * ? “ < >.

Data Repository

Click the **Open** button and navigate to a folder where your project data will be saved. This folder can be on your local hard drive, a mapped network drive or external device. You must have full access (read and write) to this folder.



Once a project is created, and data is added or created, various files and folders are created in the data repository. The **Data Repository** folder structure is outlined in the image below:



Note: The **NumericalModels** subfolder will be generated only after you have translated your conceptual model to a numerical model.

Note: The project file name should never be manually renamed, *i.e.*, using Window's Explorer. If you wish to use a different name for the project file, please use the **Save As** option (**File > Save As**).

Description (*Optional*)

This field allows you to enter optional information about your project. You can edit this information once your project has been created by clicking on **File > Project Settings..** from the main menu.

Project Coordinates

Specify the coordinate system of the project from the **Coordinate System** combo box. Hydro GeoBuilder supports the following coordinate systems:

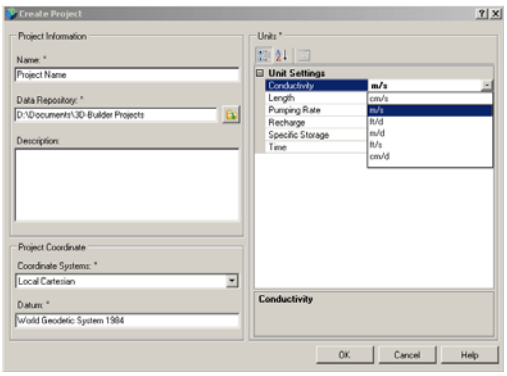
- Projected Coordinate Systems
  - State Plane 27, State Plane 83
  - UTM WGS 72, UTM WGS84, UTM NAD27, UTM NAD83
- Local Cartesian

Your project **cannot** be created using a geographic coordinate system, *i.e.*, using latitude and longitude in decimal degrees for the X-Y coordinates. However, you can import data that is in geographic coordinates. In this case, Hydro GeoBuilder will

perform a geotransformation on the coordinates so that they are expressed in the projected coordinate system of the project.

Units

Specify the default project units for each parameter using the combo boxes in the **Unit Settings** grid.



Each of the parameters provides a selection of both Imperial units and SI units.With Hydro GeoBuilder, the units **do not** need to be consistent between SI and Imperial, such that a model may have hydraulic conductivity values assigned in units of cm/sec and well pumping rates in U.S. GPM or GPD.

During data import (see Chapter 3: Importing Data), if the specified units of the data being imported are different than those specified in the **Unit Settings**, Hydro GeoBuilder will automatically convert the imported values to be expressed in the default project units.

2.2 Modifying Project Settings

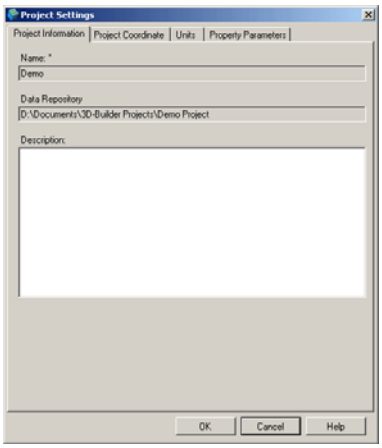
Once your project has been created, you may modify the project settings.

- From the main menu, go to **Project > Project Settings...**

The **Project Settings** dialog contains the following tabs: **Project Information**, **Project Coordinates**, **Units** and **Property Parameters**. Each tab is described below.

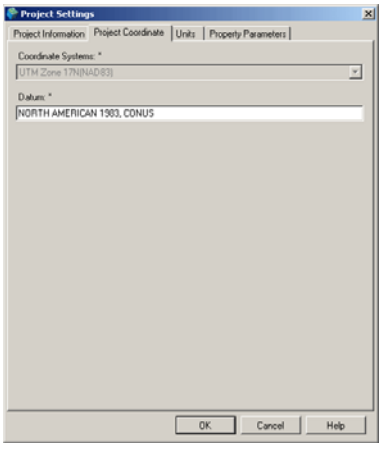
Project Information

The Project Information tab (shown above) allows you to modify the project **Description**. The **Project Name** and **Data Repository** cannot be modified.



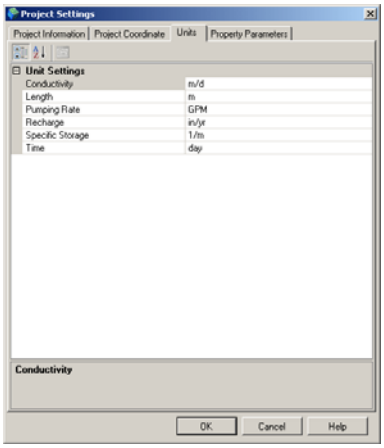
Project Coordinate

The **Project Coordinate** tab allows you to view the coordinate system and datum of the project. This information is read-only and cannot be modified.



Units

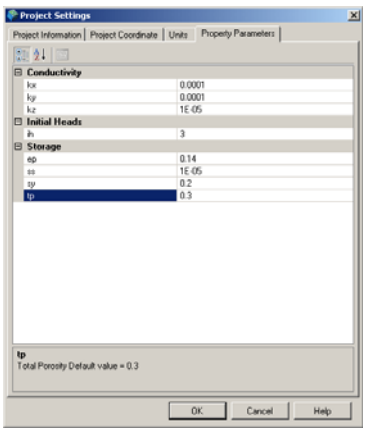
The **Units** tab allows you to view and modify the default project units.



Warning! Changing the default units only changes the unit labels. Model parameter values are NOT automatically converted to the new units settings.

Property Parameters

The **Property Parameters** tab allows you to view and modify the default property parameter values. During model translation, these parameters will be assigned to all areas of the conceptual model that have not been assigned property parameters by defining property zones (see Chapter 9: Property Modeling).



2.3 Saving a Project

It is good practice to regularly save your project to avoid any accidental loss of data. To save your project, follow the steps below:

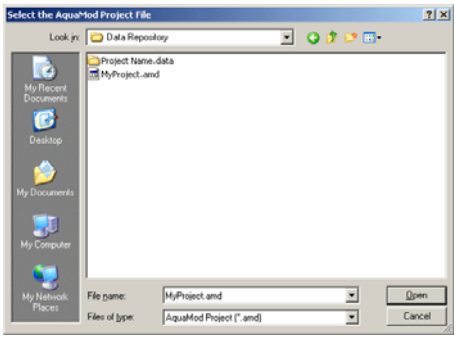
- From the main menu, select **File** > **Save** to save it using the current project name, or click the **Save As** button to save it using a different name and location.

Alternatively, you can save your project by clicking the **Save** button from the tool bar.

2.4 Opening a Project

To open an existing project, follow the steps below:

- From the main menu, select **File > Open Project... .** The **Open** dialog will display.



- Navigate to the folder that contains the ***.amd** project file, select it and then click the **[Open]** button.

Note: The original data structure of the project's data repository must be preserved in order to open a saved project. In other words, a project will fail to open if the project file or data files have been deleted or moved out of the data repository. Please see "Data Repository" on page 16 for more information.

3

Importing Data

Introduction

Hydro GeoBuilder supports importing data from various standard data types to allow you flexibility in constructing your conceptual model. Data can be imported and used in several ways; spatial data can be used to delineate and visualize geometry of structural zones, horizons and features of your conceptual model, while attribute data can be used in assigning properties to structural zones and attributes to boundary conditions.

Please see “Appendix A: Supported Data Types” on page 207 for an overview the supported data types and their uses in Hydro GeoBuilder.

About the Import Process

The data import process in Hydro GeoBuilder varies slightly depending on the data type being imported. However, in general, the importing process consists of the following steps:

[1] Select the data type and source file

[2] Specify the coordinate system and datum of the source data

[3] Map the source data fields with required target fields, and optionally create attribute fields

[4] Data preview and validation

The following sections provide additional information on the import process for each data type:

- Importing Points (page 24)
- Importing Polylines (page 30)
- Importing Polygons (page 32)
- Importing Surfaces (page 33)
- Importing Wells (page 34)
- Importing 3D Gridded Data (page 41)
- Importing Cross Sections (page 44)
- Importing Maps (page 46)

- Importing Time Schedules (page 53)

About Data Objects

After the import process, a **Data Object** is added to the **Data Explorer**. In Hydro GeoBuilder, a **Data Object** refers to any data set or data element that has been imported, or created manually using the 2D editing tools. From the **Data Explorer**, you can view the data in a 2D or 3D Viewer, view and modify data properties, and perform arithmetic and geometric operations on the data (see Chapter 4 for more information).

All imported data is stored in the **[projectname].data** folder in the data repository.

3.1 Importing Points

Points represent discrete locations in space (XYZ) where attribute information is known. Examples of points data include: ground surface or subsurface elevations, well tops, locations with known aquifer hydraulic properties, etc. Typically, this information may come from drilling wells or monitoring events where information is gathered from a specific location.

Once imported, points data can then be interpolated to generate surfaces. These surfaces can be used to create conceptual model horizons, or for defining property values for structural zones. For more information on creating surfaces from points data please see Chapter 6: Creating Surfaces.

Hydro GeoBuilder supports the following file types for points data:

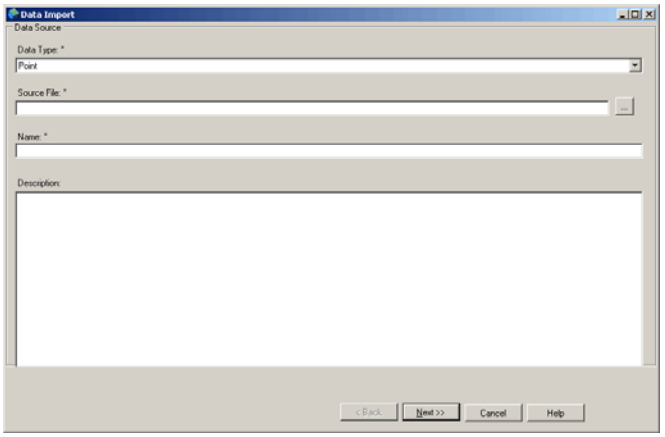
- Shapefile, *.**SHP**
- AutoCAD, *.**DXF**
- Text, *.**TXT**
- Comma-Separated Values, *.**CSV**
- Access, *.**MDB**
- Excel, *.**XLS**

For **Points** data, the following data must exist in separate columns, in your source data:

- **X**
- **Y**
- **Elevation**
- **Attribute 1 (optional) (e.g., conductivity)**
- **Attribute 2 (optional) (e.g., layer 2 elevation)**
- **etc..**

To import points data, follow the steps below:

- Right-click in the **Data Explorer**, and select **Import Data...** from the pop-up menu. The import dialog will open:

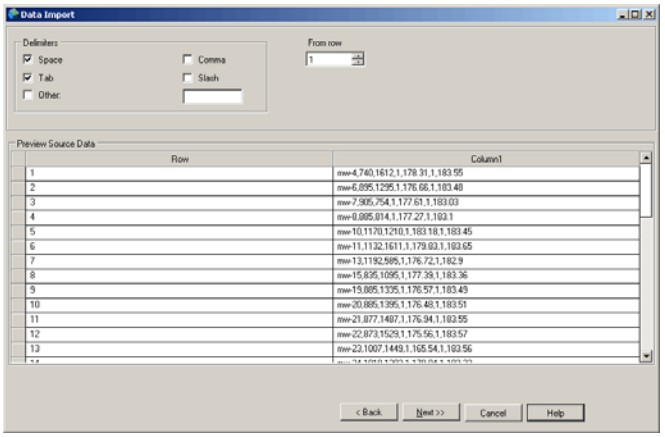


Select **Points** from the **Data Type** drop-down list. Click the **[...]** button and locate the source file.

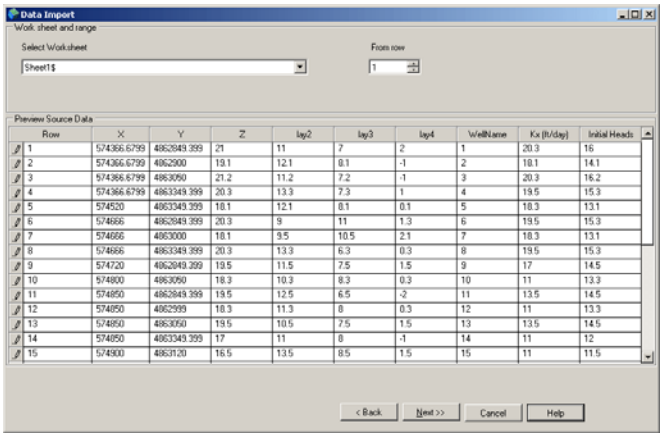
Enter a **Name** and a **Description** (*optional)* for the imported data, and click **[Next]** to continue.

- The next step allows you to preview the source data before importing, and will vary depending on which file type is selected.

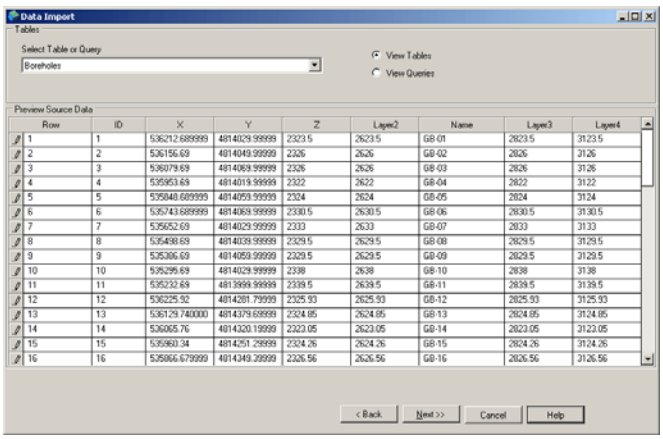
For .**CSV** and **.TXT** files (shown below), select the appropriate delimiter from the **Delimiters** frame, *e.g.*, if it is a *.CSV file, you would select “comma”. Specify which row to start importing from using the **From row** selector.



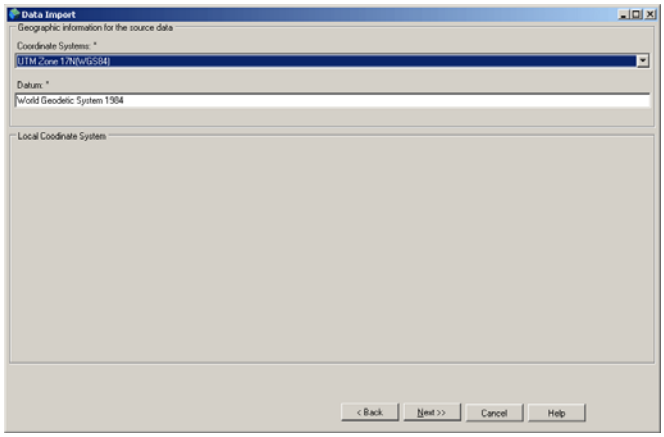
For **.XLS** files (shown below), select which Excel worksheet to import from the **Select Worksheet** drop down list. Also, you can choose which row to import from using the **From row** selector.



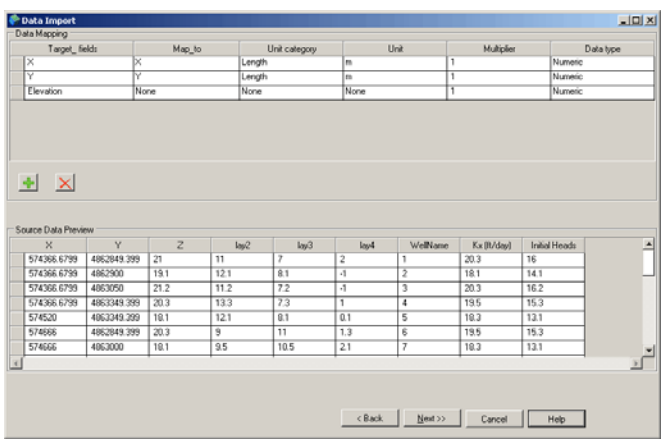
For **.MDB** files (shown below), you can choose to import data from a table or a query, by selecting either the **View Tables** or **View Queries** radio button. Select the desired query/table from the **Select Table** or **Query** drop down list box.



- Next, select the **Coordinate System** of the data being imported. If the coordinate system is different than the one defined in the project settings, Hydro GeoBuilder will perform a geotransformation, converting all coordinates to the project's coordinate system. Click the **[Next>>]** button to continue to the next step.



- Next you set your **Data Mapping** by mapping columns in the source data to the target fields in Hydro GeoBuilder. A read-only preview of the source data is presented. The process of data mapping is described in the following section in greater detail.

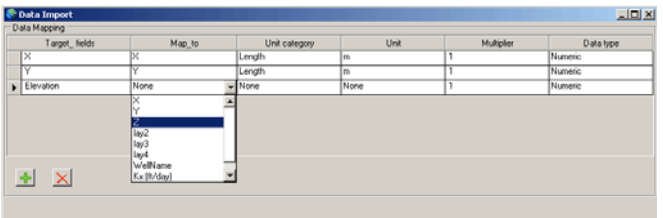


Data Mapping

The first column in the **Data Mapping** table, named **Target_Fields**, contains the required target fields for the data object. The second column, named **Map_to**, allows you to match the fields in the source data to each required target field.

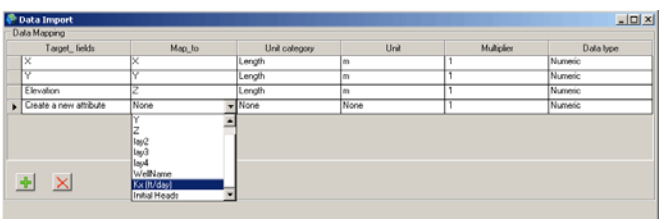
If the column labels in the source data are identical to the labels of the target fields, Hydro GeoBuilder will automatically map the columns for you. However, if the labels differ, you must map the columns manually.

To map a source field to a target field, select the corresponding source field from the drop list box in the **Map_to** column. The drop down list displays the column headers in the data source file.



For example, in the figure above, the elevation field in the source data is labeled “Z”. To map this field to the target field “Elevation”, select “Z” from the adjacent drop down list.

Source fields that are not required, can be mapped by creating a new attribute. To create a new attribute, click the **Add a new attribute** button. A new row will be added to the **Data Mapping** table.



In the **Map_to** column, select the desired attribute field in the source data, from the combo box. Repeat for additional attributes. You can delete a mapped attribute by selecting the row from the **Data Mapping** table, and then clicking the **Delete** button.

Use the **Unit Category** and **Unit** columns to define the units of a mapped field. If the specified units are different than those defined in the **Project Settings**, Hydro GeoBuilder will automatically convert the data in the source file to the default project units.

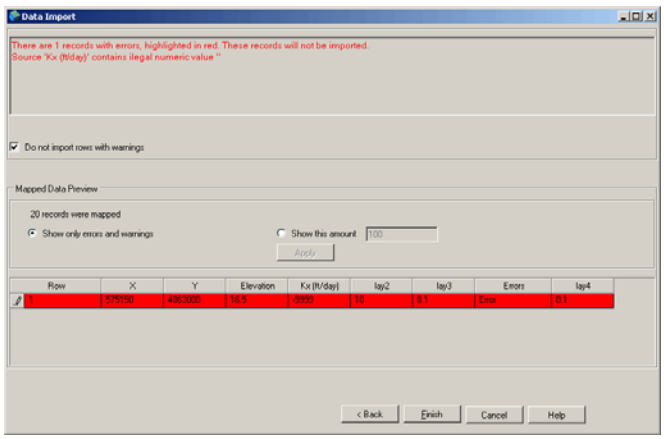
The **Multipler** column allows you to multiply all values in the mapped field by a specified multiplier value.

The **Data Type** column allows you to define the data type. Select from the following options: **Numeric**, **Text**, **Boolean**, **Date** and **Time**. For example, if the mapped column contains text data, select **Text** from the drop down list.

Once the data mapping is complete, click the **[Next]** button to continue to the validation dialog.

Data Validation

- The final step involves validation of the data being imported. This step will ensure that the data set contains valid data for each of the mapped fields.



In the top half of the dialog, Hydro GeoBuilder will list any mapped fields that contain invalid data, along with a reason for why they are deemed invalid. The data validation rules for each mapped column are as follows:

- X and Y values must be a numeric value, and present in each row of the mapped data
- Data columns will be deemed invalid if Hydro GeoBuilder detects a null (blank) field
- Data in each column must satisfy the specified data type.

If invalid data exists, you can choose to import this data anyway. Otherwise, you can select the **Do not import rows with warnings** check box, and Hydro GeoBuilder will not import any rows deemed invalid.

In the bottom half of the dialog, there are two options:

- **Show only errors and warnings:** When selected, only the records deemed invalid will be shown in the preview table. Records that are deemed invalid will be colored either red (error) or yellow (warning).
- **Show this amount:** When selected, you can view a specified number of

records in the preview table below. Enter a value, and then click the **[Apply]** button to show the records (both valid & invalid) in the preview table.

Click the **[Finish]** button to import the data. Once imported, a data object will be added in the **Data Explorer**.

3.2 Importing Polylines

Polyline data consists of a series of points (vertices) connected by lines. Polyline data objects can be used in Hydro GeoBuilder for defining geometry and assigning attributes to linear boundary conditions, such as River and Drain boundary conditions. Polylines may also be useful to visualizing geographic features such as river and road networks.

Hydro GeoBuilder supports the following file types for polyline data.

- Shapefile, *.**SHP**
- AutoCAD, *.**DXF**

To import polyline data, follow the steps below:

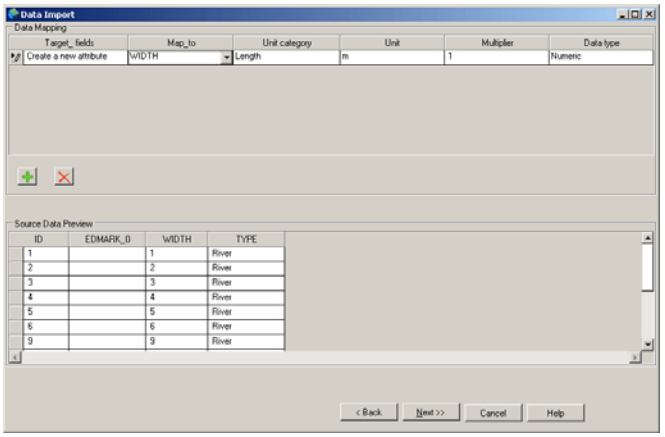
- Right-click in the **Data Explorer**, and select **Import Data...** from the pop-up menu.
  - Select **Polyline** from the **Data Type** drop down list.
  - Click the **[...]** button and locate the source file.
  - Enter a **Name** and a **Description** and then click **[Next]** to continue.
- The next step involves selecting the coordinate system of the source file, and will vary depending on which file-type is selected for the source file.

For .SHP files, if the associated .PRJ file is located in the same location as the source file, Hydro GeoBuilder can automatically detect the coordinate system of the source data, and will perform a geotransformation if the coordinate system is different than that defined in the project settings. If a .PRJ file is missing, than you will be prompted to select the **Coordinate System** for the data being imported.

For .DXF files. you will always be prompted to select the **Coordinate System** of the selected source file.

Click the **[Next]** button to continue.

- If the file type is **.SHP**, the next step involves creating attributes. If you are importing from **.DXF** file, you can skip this step.



This dialog allows you to import shapefile attributes. To create a new attribute, click the **Add a new attribute** button. When selected, a new row will be added to the **Data Mapping** table.

In the **Map_to** column, select the desired attribute field in the source data, from the combo box. Repeat for additional attributes. You can delete a mapped attribute by selecting the row from the **Data Mapping** table, and then clicking the **Delete** button.

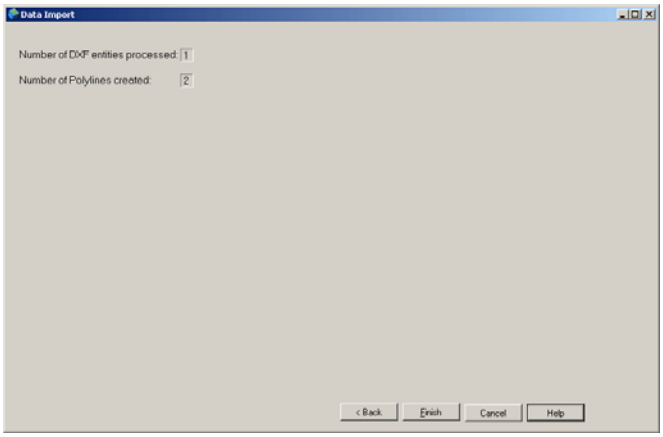
For a description of the **Unit Category**, **Unit**, **Multiplier** and **Data Type** columns, please refer to section “Data Mapping” on page 27.

Click the **[Next]** button to continue.

- The final step involves validation of the data being imported. This step will ensure that the data set contains valid data for each of the mapped columns.

For **.SHP** files, please refer to “Data Validation” on page 29 for more information on the data validation step.

For **.DXF** files, the following dialog will show, indicating the number of polylines that will be created from the source file.



Click the **[Finish]** button to complete the polygon importing process. Once imported, a polyline data object will be added to the **Data Explorer**.

3.3 Importing Polygons

Polygons are closed shapes consisting of vertices, line segments and have at least 3 sides. Polygons can be used in Hydro GeoBuilder in the following ways:

- To define the horizontal boundary of a conceptual model
- To define the geometry and attributes of horizontal boundary conditions, *e.g.*, recharge, specified-head.
- To define the geometry and attributes of property zones.
- To visualize spatial variation of geographic features using various style settings.

Hydro GeoBuilder supports the following file types for polygon data.

- Shapefile, *.**SHP**
- AutoCAD, *.**DXF**

To import polygon data, follow the steps below:

- Right-click in the **Data Explorer** and select **Import Data...** from the pop-up menu.
    - Select **Polygon** from the **Data Type** drop down list box.
    - Click the **[...]** button and locate the source file.
    - Enter a **Name** and a **Description** for the imported data, and click **[Next]** to continue.

The remaining workflow for importing polygon data is very similar to that of importing Polylines. For more information on how to import polygons, please refer to “Importing Polylines” on page 30.

3.4 Importing Surfaces

Surface data consists of an ordered array of interpolated values at regularly spaced intervals that represent the spatial distribution of an attribute, e.g., digital elevation models. Surface data can be used in Hydro GeoBuilder in the following ways:

- To define the horizons (structural zone vertical boundaries) of a conceptual model.
- To define the spatial distribution of a boundary condition attribute.
- To define the spatial distribution of a property zone attribute, *e.g.*, conductivity, initial heads.
- To visualize the spatial variation of model features, *e.g.*, surface topography, water table elevation, etc.

Hydro GeoBuilder supports the following surface file types:

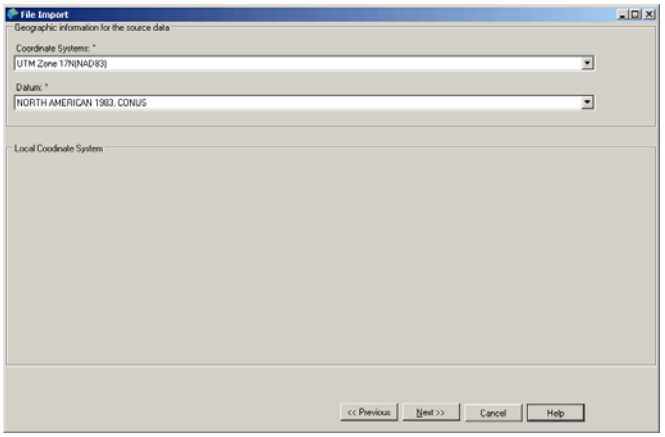
- ESRI ASCII Grid, ***.ASC, *.TXT**
- Surfer Grid , ***.GRD**
- USGS Digital Elevation Model, ***.DEM**

To import surface data, follow the steps below:

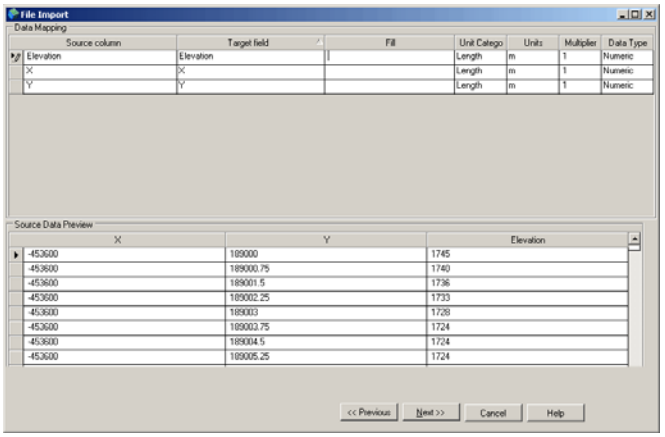
- Right-click in the **Data Explorer** and select **Import Data...** from the pop-up menu.
    - Select **Surface** from the **Data Type** drop down list box.
    - Click the **[...]** button and locate the source file.
    - Enter a **Name** and a **Description** for the imported data, and click **[Next]** to continue.

Note: Surface files that contain a large quantity of data points may require substantial time to import into Hydro GeoBuilder.

- Next, select the **Coordinate System** of the data being imported. If the coordinate system is different than the one defined in the project settings, Hydro GeoBuilder will perform a geotransformation, converting all coordinates to the project’s coordinate system. Click the **[Next]** button to continue to the next step



- Surface data will usually only consist of three columns: **X**, **Y** and **Attribute Data** (elevation, conductivity, etc). Hydro GeoBuilder will automatically map the source columns to the target fields. You can preview the mapped data before importing into Hydro GeoBuilder.



Click the **[Next]** button to finish the import process. Upon importing, a new data object will be added to the **Data Explorer**.

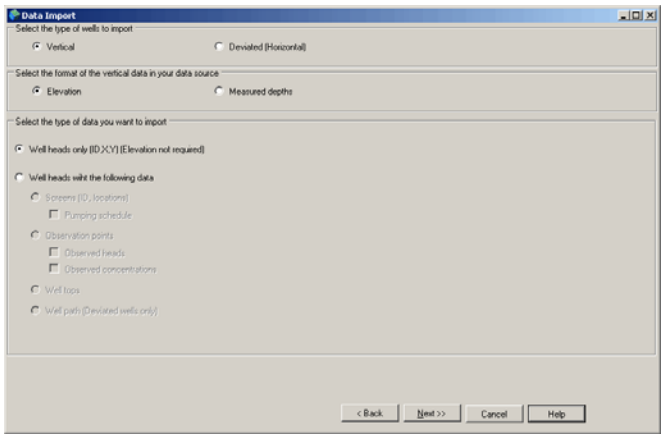
3.5 Importing Wells

Hydro GeoBuilder supports the following file types for well data.

- Excel,***.XLS**

To import wells, follow the steps below:

- Right-click in the **Data Explorer** and select **Import Data...** from the pop-up menu.
  - Select **Wells** from the **Data Type** drop down list.
  - Click the **[...]** button and locate the source file.
  - Enter a **Name** and a **Description** for the imported data, and click **[Next]** to continue.
- The next step allows you to preview the source data before importing, and choose a subset of the source data, *e.g.*, a worksheet of an excel file. Select which Excel worksheet to import from the **Select Worksheet** drop-down list. Also, you can choose which row to import from using the From Row text field.
- Hydro GeoBuilder provides you with various options for importing wells, and associated well data. For example, you can choose to import well heads (Well ID, X, Y, Elevation, Bottom), or you can choose to import wells heads along with related screen locations, pumping schedules, or observation points.



In the **Select the type of wells to import** frame, choose between **Vertical** and **Deviated (Horizontal)**.

Note: For **Deviated (Horizontal) Wells,** only well heads and well paths can be imported via the import process. Wells screens, observation points and well tops for horizontal wells can be defined later in the Wells table. For more information on the well table, please see “Well Table” on page 65.

Next, specify how the vertical data is expressed in the source file. If the data is expressed with respect to a vertical reference datum, *e.g.*, above mean sea level, then select **Elevation**. If the vertical data is expressed as a depth, *e.g.*, distance from ground to the bottom of the well, then select **Measured Depth**.

Next, select the type of vertical well data to import (horizontal wells not supported). Select the **Well Heads Only** option to import just the well heads, *i.e.*, X-Y coordinates, elevation, and well depth for each well. Select the **Well Heads with the Following Data** option to import additional data for each well. Options include:

Note: For data requirements for each option please see the next step “Data Mapping”

Screen ID Location: Import the screen locations for each well. Select the **Pumping Schedule** check box to also import related pumping schedules for each screen. This data could be used later to define pumping well boundary conditions.

Observation Points: Import observation points for each well. Select either **Observed heads**, **Observed Concentrations**, or both.

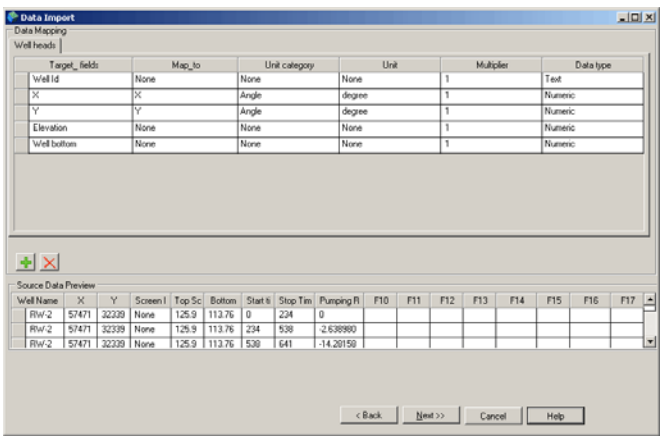
Well tops: Import the elevation (or measured depth) of points along the well path, where formation tops (horizons) intersect with the well. This data could be used later to generate surface and horizon layers.

Well Paths: Please see “Well Heads with Well Path” on page 40.

Once you have selected which well data to import, click the **[Next]** button to proceed to the data mapping .

- This step requires you to map the columns in the source data to the required target fields. The required fields will vary depending on the type of well data you selected in the previous step. The following sections describe the data mapping for each data type option:

Well Heads Only



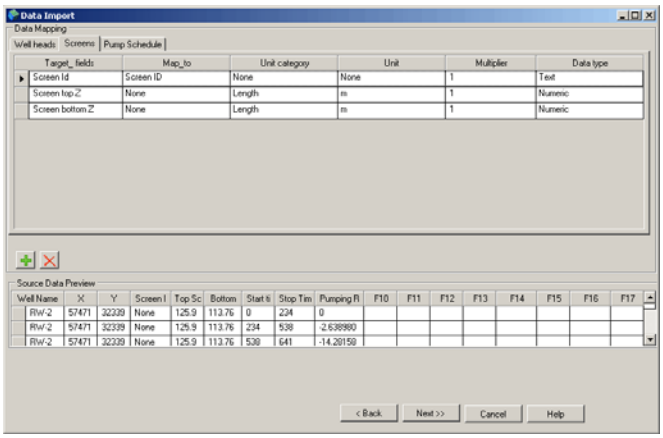
For importing **Well Heads** only, you must map the following columns from the source data to the required target fields:

- Well ID

- X
- Y
- Elevation
- Bottom

Well ID must be a unique value in the source data. If not, any rows containing duplicate Well IDs will not be imported.

Well Heads with Screens



If this option is selected, you must first map the well heads under the **Well Heads** tab. Next, click on the **Screens** tab, and map the appropriate columns from the source data to the following target fields:

- Screen ID
- Screen Bottom Z (elevation of bottom of screen)
- Screen Top Z (elevation of top of screen)

For each well in the source data, the Screen ID must be unique. Also, screens should not overlap within a single well. These requirements will be validated in the final step of the well import process.

If you selected the pumping schedule check box in the previous step, click the **Pumping Schedule** tab and then map the appropriate columns from the source field to the following target fields:

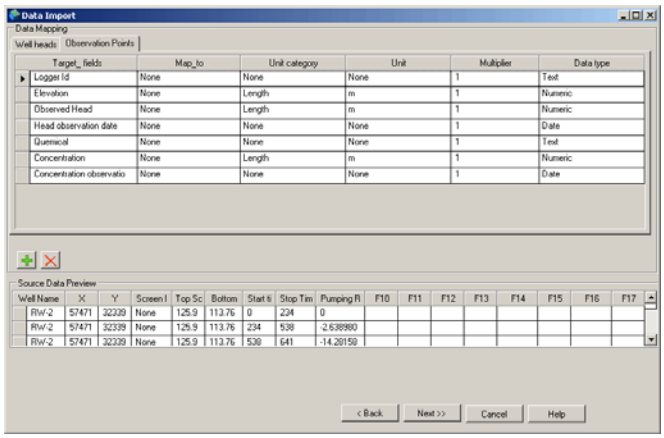
- Pumping Start Date, in *MM/DD/YYYY HH:MM:SS* format, time is optional.
- Pumping End Date, in *MM/DD/YYYY HH:MM:SS* format, time is optional.
- Pumping Rate

Note: Please consider the following when importing a pumping schedule:

- In your source data, the final time in the pumping schedule should have a pumping rate of 0 to indicate the stop time.

- If time is not included in the source data (just the date), Hydro GeoBuilder will automatically set the time to 12:00:00 pm.
- Currently, Pumping Schedules can only be imported using absolute time. Please ensure that the date and time values in your source data are expressed in absolute time (MM/DD/YYYY HH:MM:SS) and not relative time (0-10 days, 10-20 days, etc..).

Well Heads with Observation Points



When this option is selected, you must first map the well heads under the **Well Heads** tab (described above). Next, click the **Observation Points** tab, and map the appropriate columns from the source data to the following target fields:

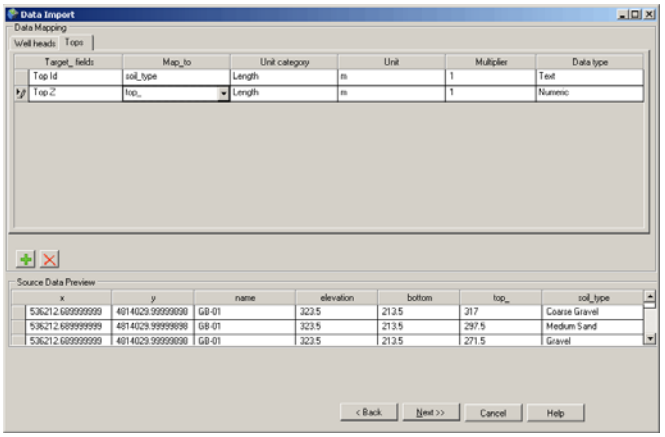
For Observed Heads:

- Logger ID
- Elevation
- Observed Head
- Head Observation Date

For Observed Concentrations:

- Logger ID
- Elevation
- Quemical
- Observed Concentration
- Concentration Observation Date

Well Heads with Well Tops



When this option is selected, you must first map the well heads under the **Well Heads** tab (described above). Next, click on the **Tops** tab and map the appropriate columns from the source data to the following target fields:

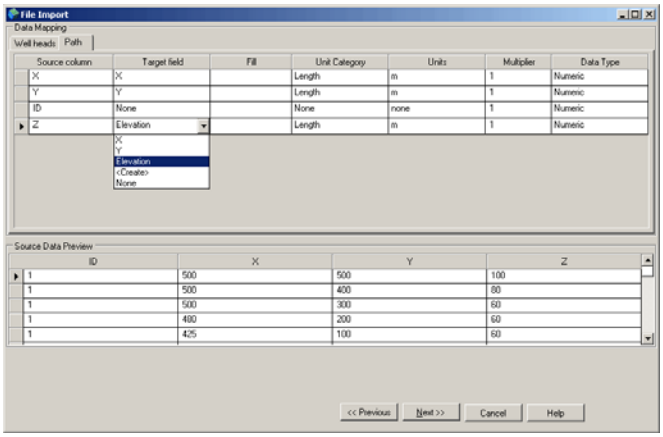
- Top Z, *elevation (or measured depth) of formation*
- Top ID, *formation name, e.g., Sand1, Sand2, Clay etc.*

Please note, the well top data in the source file must be formatted as follows:

Well ID	**Top ID**	**Top X**
Well1	Fill	6.5
Well1	Sand1	26
Well1	Clay	52
etc.	etc..	etc..
Well2	Fill	4
Well2	Sand1	17
Well2	Clay	94
etc.	etc..	etc..

Well Heads with Well Path

This option is only available for deviated (horizontal) wells.



When this option is selected, you must first map the well heads under the **Well Heads** tab (see “Well Heads Only” on page 36). Next, click on the **Path** tab and map the appropriate column from the source data to the following target field:

- Elevation
- X
- Y
- Well ID

Please note, well path data in your source file must be formatted as follows.

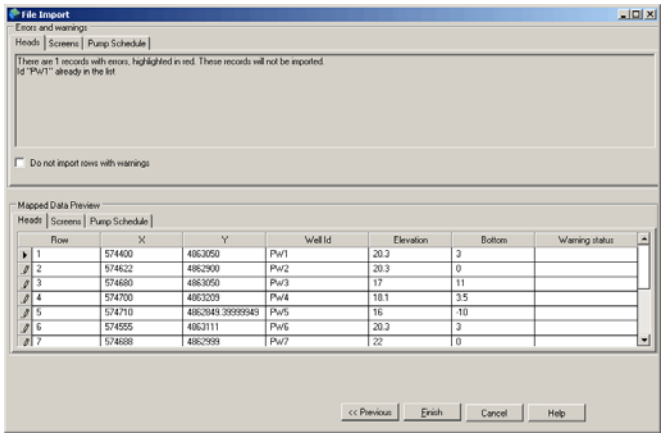
Well ID	**X**	**Y**	**Elevation**
Well1	574506.3	4863299.36	100
Well1	574506.11	4863299.36	80
Well1	574506.60	4863298.36	68
etc...	etc..	etc..	etc..

Each row in the data represents a vertex in the well path. When viewed in 3D Viewer, Hydro GeoBuilder connects each vertex with a line, allowing you to visualize the horizontal well path(s).

Once the well path is imported you can manually define screen intervals, pumping schedules, observation points and well tops in the Wells Table. Please see “Well Table” on page 65 for more information.

- The final step in the Well import process is data validation. Hydro GeoBuilder will validate the mapped data, and highlights any rows that contain invalid data,

e.g., null values, wrong assigned data type, duplicate rows etc.



Please see “Data Validation” on page 29 for more information on data validation.

Click the **[Next]** button to import the data. Once imported, a Wells data object will be added in the **Data Explorer**.

3.6 Importing 3D Gridded Data

3D Gridded Data refers to 3D grids with attributes assigned to each grid cell. 3D Gridded data can be used in Hydro GeoBuilder to visualize heads generated from a MODFLOW run in Visual MODFLOW, or for assigning spatially-variable attributes to boundary conditions and property zones. Hydro GeoBuilder supports the following file types for 3D Gridded data:

- MODFLOW Heads file**,*.HDS**
- Visual MODFLOW .DAT files, ***.DAT**

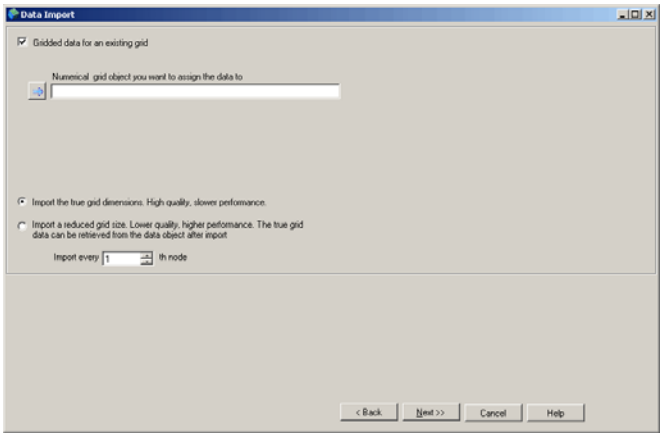
Note: In order to import data from MODFLOW .HDS files, the source file must exist in the folder that contains all associated MODFLOW data files, *e.g.*, .DIS, .NAM etc.

Note: Visual MODFLOW .DAT files must include the following data: X, Y, Z and a parameter. For information on how to export gridded data from Visual MODFLOW in the .DAT format, please refer to the Visual MODFLOW User’s Manual.

To import 3D Gridded data, follow the steps below:

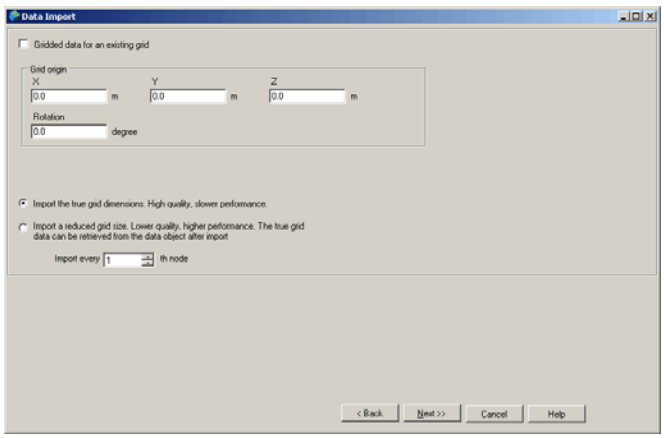
- Right-click in the **Data Explorer** and select **Import Data...** from the pop-up menu.
  - Select **3D Gridded Data** from the **Data Type** drop down list.
  - Click the **[...]** button and locate the source file.
  - Enter a **Name** and a **Description** for the imported data, and click **[Next]** to continue.

- Next, select the **Coordinate System** of the 3D Gridded data. If the coordinate system is different than the one defined in the project settings, Hydro GeoBuilder will perform a geotransformation, converting all coordinates to the project's coordinate system. Click the **[Next]** button to continue to the next step.
- The next step allows you to specify how the gridded data is to be imported.


Data Import
Gridded data for an existing grid
Numerical grid object you want to assign the data to
Import the true grid dimensions. High quality, slower performance.
Import a reduced grid size. Lower quality, higher performance. The true grid data can be retrieved from the data object after import
Import every 1 th node
< Back
Next >>
Cancel
Help

When importing a **.HDS** file, the **Gridded data for existing grid** option will be automatically selected. The grid dimensions in the source file must be identical to the dimensions of a grid in your project. Select the existing numerical grid from **Conceptual Model** tree, and then click the button.

When importing a **.DAT** file, the **Gridded data for existing grid** option will be unchecked.

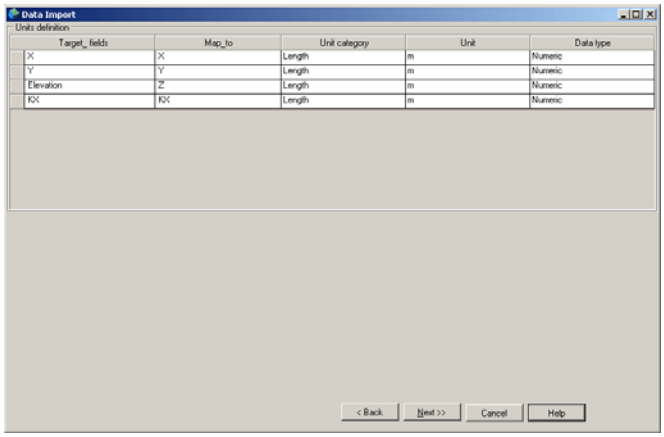


When this option is selected, the **Grid Origin** frame will display. If the gridded data is in model coordinates, specify the **Grid Origin** and the degree of **Rotation**. If the grid is in world coordinates, you may leave the grid origin as is.

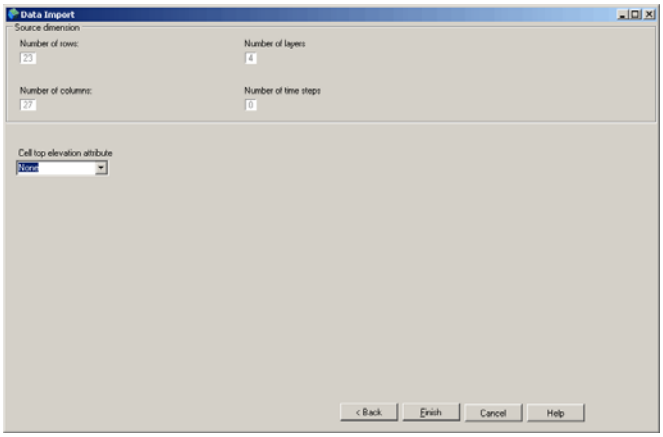
For both file types, you can specify the grid resolution. To import the full grid dimensions, select **Import the true grid dimensions** option. Please note that depending on the performance capabilities of your computer, 3D Gridded data containing large volumes of data may take a significant time to import.

To improve importing and viewing performance, select the **Import a reduced grid size** option, and specify a value in the **Import every nth node** box. For example, if a value of 2 is defined, then Hydro GeoBuilder will only display every other node in the 3D grid.

- Next, select the appropriate **Data Category**, **Unit** and **Data Type** for each of the mapped attributes.



- The final dialog in the import process for 3D Gridded shows the grid dimensions of the source data.



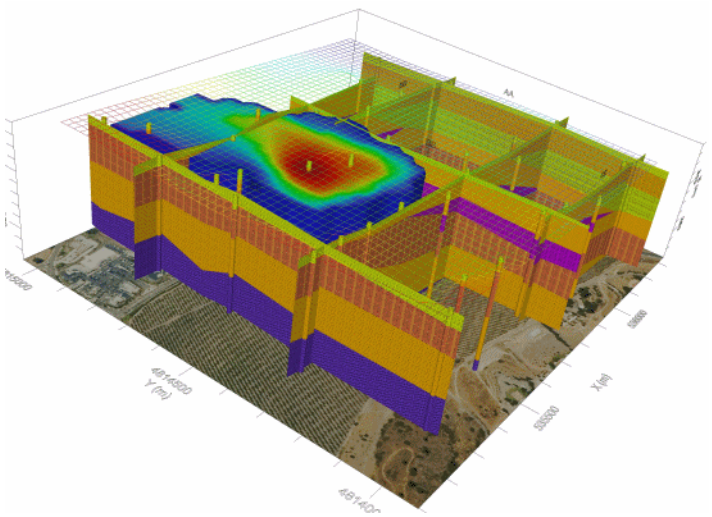
The **Source Dimensions** frame displays the **Number of Rows**, **Number of Layers**, **Number of Columns** and **Number of Time Steps** in the source data.

Finally, click the **[Finish]** button to import the 3D Gridded data.

3.7 Importing Cross Sections

Hydro GeoBuilder is capable of importing 3D cross sections generated by Hydro GeoAnalyst (HGA) data management software.

For information on how to create 3D cross section in HGA, please refer to the HGA User's Manual. For HGA product information, please visit our website: www.swstechnology.com or contact your Schlumberger Water Services sales representative.



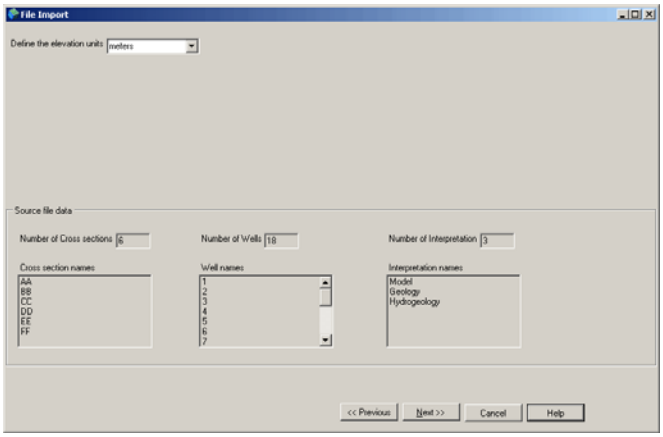
When a cross section is created in HGA's 3D Explorer, a file (*.3XS) is saved in the **v3D** folder, located in the HGA project folder. By default, the location of this folder is,

C:\Program Files\HGAnalyst\Projects\[Project Name]\v3D

A *.3XS file contains information on the wells and layers of each cross section. The cross section can contain geology, hydrogeology, and model layer interpretation layers. When imported into Hydro GeoBuilder, this data can be used for generating surfaces and horizons from interpretation layers, or simply for visualization purposes.

To import a cross section file in Hydro GeoBuilder, follow the steps below:

- Right-click in the **Data Explorer** and select **Import Data...** from the pop-up menu.
  - Select **Cross Section** from the **Data Type** drop down list.
  - Click the [...] button and locate the source file.
  - Enter a **Name** and a **Description** for the data, and click **[Next]** to continue
- Next, select the **Coordinate System** of the cross section. If the coordinate system is different than the one defined in the project settings, Hydro GeoBuilder will perform a geotransformation, converting all coordinates to the project's coordinate system. Click the **[Next]** button to continue to the next step.
- The final step involves selecting the elevation units and previewing the cross section data.



At the top of the dialog, specify the cross section elevation units from the drop-down list box. You can choose from **metres** or **feet**.

The **Source File Data** frame contains information about the selected source file. Here you can preview the **Number of Cross sections** in the source file, along with the cross section names. The **Number of wells** in the source data is shown, along with the well names. Finally, the number and type of interpretations in the source data are shown, *e.g.*, Model, Geology and HydroGeology.

Click the **[Finish]** button to import the cross section data. Upon importing, a new cross section data object will be added to the **Data Explorer**.

3.8 Importing Maps

Site maps of the model region, such as aerial photographs, topographic maps and satellite imagery, are often useful for gaining a perspective of the dimensions of the model, and for locating important characteristics of the model. Although maps do not contain any specific data used in the calculations, and the presence of a map does not influence the results of the simulation, they are useful for enhancing visualization of the model.

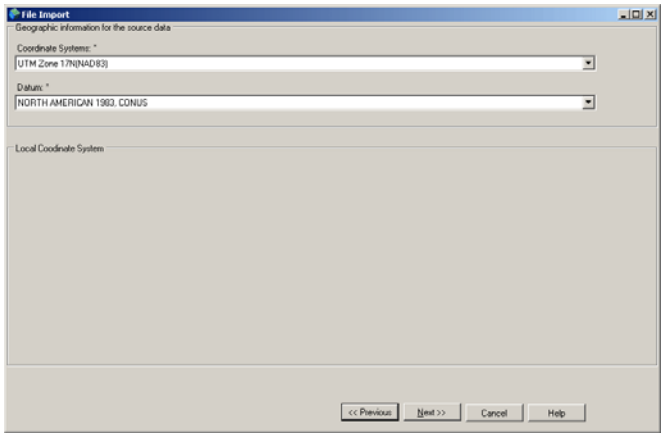
Hydro GeoBuilder supports the following raster graphics file types:

- ***.BMP,** *Bitmap*
- ***.TIF,** *Tagged Image Format*
- ***.JPG,** *JPEG Interchange Format*

Note: When a raster image is imported into Hydro GeoBuilder, the source file is copied and saved in the project's data repository folder. As such, the original file may be modified, moved or deleted without affecting the imported raster image.

To import a map into Hydro GeoBuilder, follow the steps below:

- Right-click in the **Data Explorer** and select **Import Data...** from the pop-up menu.
  - Select **Map** from the **Data Type** drop down list.
  - Click the **[...]** button and locate the source file.
  - Enter a **Name** and a **Description** for the data, and click **[Next]** to continue
- Next, select the **Coordinate System** of the image file. If the coordinate system is different than the one defined in the project settings, Hydro GeoBuilder will perform a geotransformation, converting all coordinates to the project's coordinate system.



Click the **[Next]** button to continue to the next step.

- All raster images must be georeferenced before importing into Hydro GeoBuilder. If the selected raster image has already been georeferenced it should have an associated georeferencing tag file and does not need to be georeferenced in Hydro GeoBuilder. Please note that the georeferencing tag file must be located in the same folder as the selected source file, in order for Hydro GeoBuilder to recognize it.

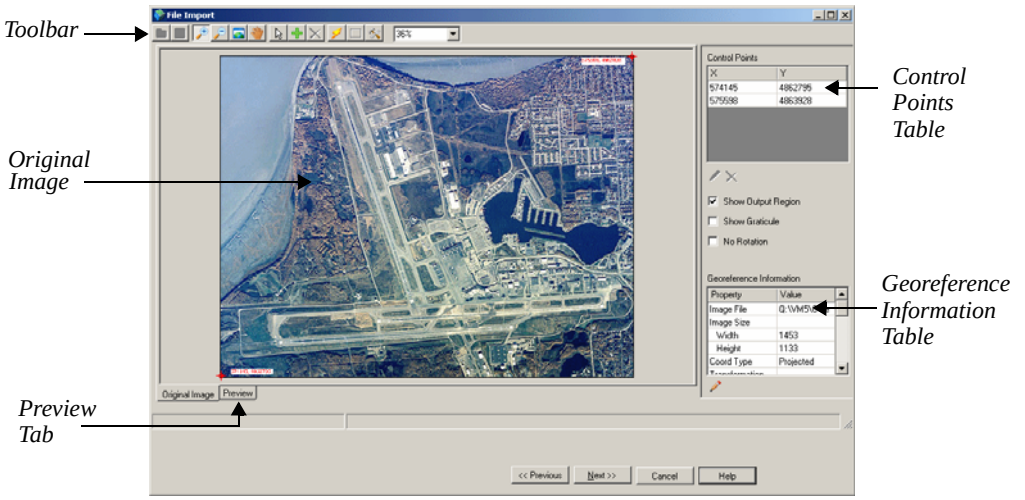
The following table summarizes the supported graphics file types, and the corresponding georeferencing tag files:

Raster Source	Georef. Tag File
*.BMP	*.BPW
*.TIF	*.TFW
*.JPG	*.JPW

If your source file has a georeferencing tag file, you can skip to the final step on page page 53. If the source file does not have a georeferencing tag file, you will be prompted to manually georeference the raster image. This procedure is described in the following section.

3.8.1 Georeferencing Images

Georeferencing a graphics file involves mapping a coordinate system to the individual pixels of the image. When this is required, the following window will display, when importing a raster image:



Toolbar Buttons

The buttons in the top toolbar are described below:

Save: Saves the transformed image, along with a corresponding georeferencing tag file.

Zoom In: Click-and-drag the mouse to select the zoom area.

Zoom Out: Zoom out of the image.

Full Extent: Zoom completely out so that the entire image is shown.

Pan: When zoomed in, move the image left, right, up or down.

Add (Control Point): Add a georeference point. See “Adding Georeference Points” on page 49.

Delete (Control Point): Delete a selected georeference point.

Transform Image: Assigns coordinates to image pixels using the specified control points.

Output Region: This button allows you to save a specified area of the image to the georeferenced file, after the image has been transformed. By default, the output region is the whole image. Click-and-drag a rectangular box on the image to define a new output region, and then click the **Save** button to save the output region to the georeference file.

Configure Georeferencing Options: Opens the configure georeferencing options dialog box. For more information on these options, please see “Configure Georeferencing Options” on page 50.

100% **Magnification Selector:** Select a magnification level from the combo box.

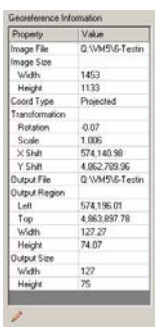
Control Points Table

The control points table contains the specified control points. You can edit an existing control point by selecting the point from the table, and then clicking the **Edit** button. A dialog box will display prompting you to modify the control point coordinates.

You can also delete a control point from the control points table. To do so, select an existing control point from the grid, and then click the **Delete** button.

Georeference Information Table

The georeference information table displays information about the georeferenced image including the original image file name and path, the original image size, and coordinate type (projected, local or geographic). It also displays information on the transformation such as the degree of rotation, scale X-Y shift, and the output file name path and image size.



Adding Georeference Points

In order to map pixels of the image to a coordinate system, the image must have at least two georeference points with known world coordinates.

To set a georeference point,

- From the top toolbar, click on the **Add** button
- Click on a map location where the world coordinates are known.
- A georeference point window will appear prompting for the X and Y world coordinates of the selected location.
- Enter the X and Y coordinates for this point.
- Repeat this procedure for additional georeference points.

When you set a georeference point, it is added to the **Control Points Table**.

You can improve the accuracy of the georeferencing by adding more than two control points to the image. When the image is transformed, the **Preview** tab will display the original control points and the corresponding georeferenced points, thus allowing you to visualize the accuracy of the georeferencing.

Once you have set at least two georeference points, click the **Transform** button to georeference the image. The georeferenced image will then be displayed in the **Preview** tab.

Editing Georeference Points

To edit a georeference point,

- Select the georeference point from the **Control Points Table**
- Once selected, click the **Edit** button (located just beneath the control points table)
- A Georeference point window will appear prompting for the X and Y world coordinates of the selected location.
- Enter the new X and/or Y coordinates for this point.
- Click the **[Ok]** button.

Note: You must click the **Transform** button again in order for the georeferencing to update to reflect the modified X-Y values.

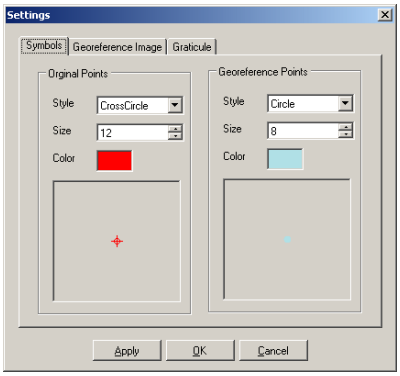
Deleting Georeference Points

To delete a georeference point,

- Select the georeference point from the **Control Points Table**.
- Once selected, click the **Delete** button (located just beneath the control points table)

Configure Georeferencing Options

When the **Configure Georeferencing Options** button is selected, the following dialog will open:

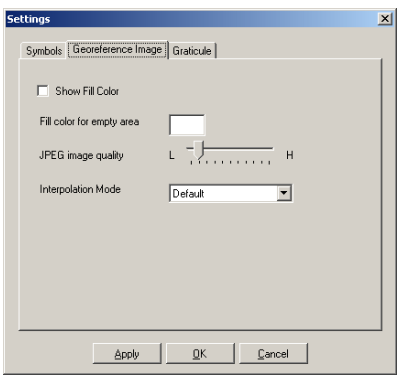


Symbols Tab

This tab allows you to change the style settings of the original control points and the georeference points. Choose a **Style**, symbol **Size** and **Color**. A preview of the symbol settings is shown in the boxes below.

Georeference Image Tab

This tab allows you to define settings for the georeferenced image. Each setting is described below.



When a georeferenced image is rotated, you can fill the areas of empty space with a specified color. Otherwise, leave the check box unchecked and the empty space will show transparent.

Click the color box beside **Fill color for empty area** to select the color to fill the empty spaces (only if **Show Fill Color** option is selected).

Use the slider to set the **JPEG Image Quality**. When set closer to **L** (low), more compression is used in the saved georeference image file, resulting in a smaller file and poorer quality. When set close to **H** (high), less compression is used in the saved georeferenced image file, resulting in a larger file with better quality.

From the **Interpolation Mode** combo box, specify the interpolation method (algorithm) to use when the image is scaled or rotated The various interpolation methods are briefly described below. Keep in mind, lower-quality interpolation mode will result in a smaller output file, whereas high-quality interpolation modes will result in a larger output file.

Default: default interpolation mode.

LowQuality: a low-quality mode.

HighQuality: a high-quality mode.

Bilinear: Bilinear interpolation. No prefiltering is done. This mode is not suitable for shrinking an image below 50 percent of its original size.

Bicubic: Bicubic interpolation. No prefiltering is done. This mode is not suitable for shrinking an image below 25 percent of its original size.

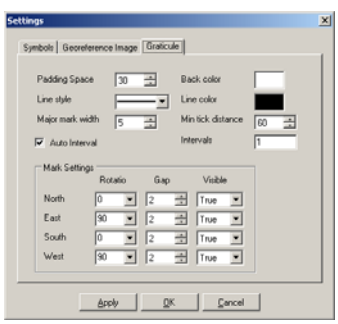
NearestNeighbor: Nearest-neighbor interpolation.

HighQualityBilinear: Specifies high-quality, bilinear interpolation. Prefiltering is performed to ensure high-quality shrinking.

HighQualityBicubic: High-quality, bicubic interpolation. Prefiltering is performed to ensure high-quality shrinking. This mode produces the highest quality transformed images.

Graticule Tab

The graticule tab provides display options for the preview graticule. These settings are described below:



Padding Space — Controls the amount of buffer space between the edge of the preview window display, and the labels on the axis. It may be necessary to increase this value when the X and Y coordinates contain many digits.

Line Style — Select from various line styles, *e.g.*, solid, dashed, etc.

Line Color — Set the color of the graticule lines

Back color	Set the background color of the graticule
Major mark width	Set the width of the major mark ticks
Minor tick distance	Set the distance between minor ticks
Auto Interval	Automatically calculates the distance between graticule lines
Interval	If Auto Interval is not selected, set the distance between graticule lines.
Mark Settings	Controls the axis labels. For each axis, you can set the visible status, rotate the label, and control the gap between the label and the axis itself.

- The final step involves previewing the raster image and viewing coordinate information, before importing into Hydro GeoBuilder.

The **Map Coordinates** frame provides the georeferenced coordinates of the **Top Right** and **Bottom Left** corners of the image. The path of the georeferenced image, and the associated georeference tag file is also shown.

Click the **[Finish]** button to import the map into Hydro GeoBuilder.

3.9 Importing Time Schedules

Time schedule data generally contains time data for one or more attributes. It can be used in Hydro GeoBuilder to define the stress periods for transient boundary condition attributes, *e.g.*, recharge, river stage etc. The following file types are supported for time schedule data:

- Excel, ***.XLS**

Time schedule data can be imported using either an absolute or relative time format. An example of an absolute time schedule is shown below:

Time	River Stage
11/01/2008	16.18
11/15/2008	16.01
12/01/2008	16.12
12/15/2008	16.29
etc..	

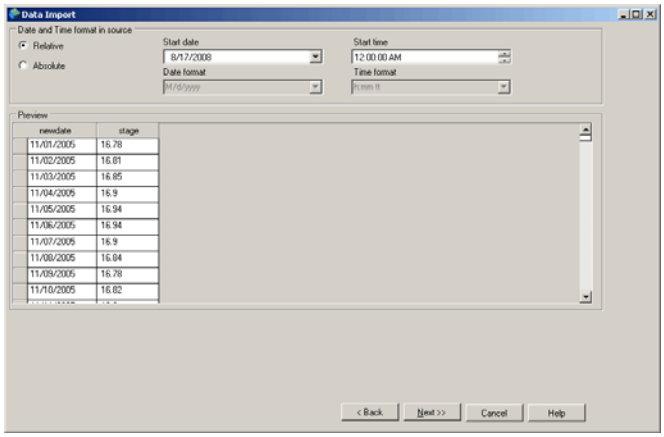
An example of a relative time schedule is shown below:

Starting Date: 11/01/2008

Time (days)	River Stage
0	16.18
15	16.01
30	16.12
45	16.29
etc..	

To import time schedule data, follow the steps below:

- Right-click in the **Data Explorer** and select **Import Data...** from the pop-up menu.
    - Select **Time Schedule** from the **Data Type** drop down list.
    - Click the **[...]** button and locate the source file.
    - Enter a **Name** and a **Description** for the imported data, and click **[Next]** to continue.
- The next step allows you to preview the source data before importing, and choose a subset of the source data, *e.g.*, a worksheet of an excel file. Select which Excel worksheet to import from the **Select Worksheet** drop down list. Also, you can choose which row to import from using the **From Row** text field.
- The next step involves selecting the type of time data used in the source file.

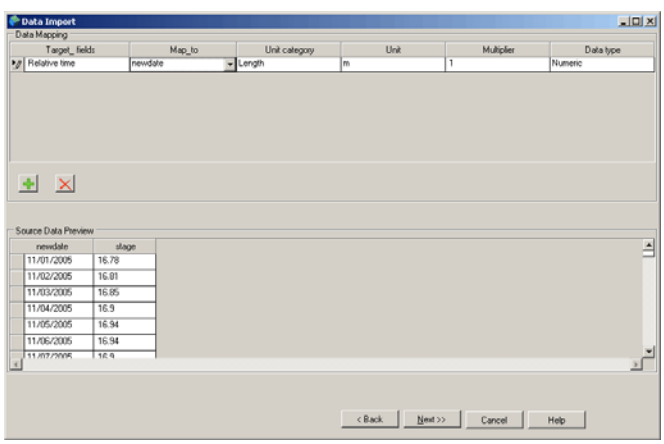


If the time values are expressed in relative time select the **Relative** option, and specify the starting date and time from the combo boxes.

If the time values are expressed in absolute time, select the **Absolute** option.

Click the **[Next]** button to continue.

- The next step involves data mapping and creating attributes. The required target fields will vary depending on which option was selected in the previous step, e.g., **Absolute** or **Relative**.



If you selected **Absolute**, the required target field will be **Start Date** and **Time**, and if you selected **Relative**, the required target field will be **Relative Time**. Map the time field in your source data to the required target field.

Next, create a new attribute, and map any associated parameter, *e.g.*, recharge, in the source data to the new attribute. If necessary, repeat for additional parameters in the time schedule.

Data mapping and creating new attributes are described in the section “Data Mapping” on page 27.

- The final step in the time schedule import process is data validation. Hydro GeoBuilder will validate the mapped data, and highlights any rows that contain invalid data, *e.g.*, null values, wrong assigned data type, duplicate rows etc.

Please see “Data Validation” on page 29 for more information on data validation.

Click the **[Next]** button to import the data. Once imported, a time schedule data object will be added in the **Data Explorer**.

3.10 Deleting Data Objects

To delete a data object, right-click on the data object from the **Data Explorer**, and select **Delete** from the pop-up menu.

3.11 Exporting Data Objects

Hydro GeoBuilder supports data export for the following data objects:

- Points, Polygons and Polylines (*.SHP, *.CSV)
- Surfaces, Horizons (*.CSV)

To export a data object, follow the steps below:

- From the **Data Explorer**, right-click on the desired data object, and select **Export** from the pop-up menu.
- A **Save As** dialog box will display on your screen
- Specify a file **Name** and **Folder** location, and the **File Type** for the exported file, and then click the **[Save]** button.
- Click the **[OK]** button from the **Export** dialog box.

3.12 Importing Data from Existing VMOD Models

Individual input elements from your existing Visual MODFLOW (VMOD) models can be exported and then imported into Hydro GeoBuilder, to assist with interpreting your results, or designing conceptual models from your VMOD models.

For information on how to import output elements from a VMOD model into Hydro GeoBuilder, please refer to “Viewing Results in Hydro GeoBuilder” on page 205.

Numerical Grid

To view the numerical grid a full 3D Object in Hydro GeoBuilder,

- From VMOD, select **Input > Grid**
- Select **File > Export > Data**
- Select **TecPLOT .DAT** format, and enter a file name.
- Select all the layers, and choose an attribute: layer top, bottom, or thickness.
- In Hydro GeoBuilder, you can import the .DAT file using the Importing 3D Gridded Data process described on page 41.

To view the numerical grid as a 2D Layer in Hydro GeoBuilder,

- From VMOD, select **Input** > **Grid**
- Select **File > Export > Image**
- Select AutoCAD.DXF format, and enter a file name.
- Select the desired layers
- In Hydro GeoBuilder, you can import the .DXF file using the Importing Polylines process described on page 30.

Layer Elevations

To export layer top or bottom elevations from VMOD, for use in Hydro GeoBuilder,

- From VMOD, select **Input > Grid**
- Select **File > Export > Data**
- Select the Surfer GRD file format
- Select the desired layers. This will generate one surface for the top (or bottom) of each selected model layer. You need to repeat these steps, in order to generate a surface for the bottom of the bottommost layer in your model.
- In Hydro GeoBuilder, you can import the .GRD files using the Importing Surfaces process described on page 33.

Properties

From VMOD, you can export the property values for the entire grid, to a 3D Gridded TecPLOT .DAT format (for example X, Y, Z, Kx). This could be useful if you want to generate new models from your existing models.

- From VMOD, select **Input > Properties**
- Select the desired parameter group (Conductivity, Storage, Initial Heads)
- Select **File > Export > Data**
- Select the **TecPLOT.DAT** file format.
- Select the desired parameters and layers.
- In Hydro GeoBuilder, you can import the .DAT file using the Importing 3D Gridded Data process described on page 41.

To export properties on a layer-by-layer basis,

- From VMOD, select **Input > Properties**
- Select the desired parameter group (Conductivity, Storage, Initial Heads)
- Select **File > Export > Data**
- Select the **Surfer GRD** file format.
- Select the desired layers and parameters.
- One file will be generated for each select layer.
- In Hydro GeoBuilder, you can import these .GRD files using the Importing Surfaces process described on page 33.
- These surfaces can be used to define property values, when you create property zones.

Wells

To export wells from VMOD, and import into Hydro GeoBuilder follow the steps below:

- In VMOD, select **Input > Wells > Pumping Wells**
- Select **Database** from the side menu.
- From the wells database table, copy the desired wells from the grid, and paste into a Microsoft Excel worksheet.
- Save the MS Excel worksheet.
- In Hydro GeoBuilder, you can import this .XLS file using the Importing Wells process described on page 34.
- The wells can be viewed in 3D or 2D and can be used to define pumping well boundary conditions.

Boundary Conditions

Boundary condition locations can be exported to a DXF or raster image:

- From VMOD, select **Input > Boundaries**
- Select the desired boundary conditions (and hide all other non-desired overlays).
- Select **File > Export > Image**
- Select the **AutoCD DXF** file format.
- In Hydro GeoBuilder, you can import this DXF file using the Importing Polylines process described on page 30.

- You may then create a new data object (polygon or polyline), and using the CAD file as a background, trace over the existing boundary conditions, to represent rivers, recharge zones, etc.

4

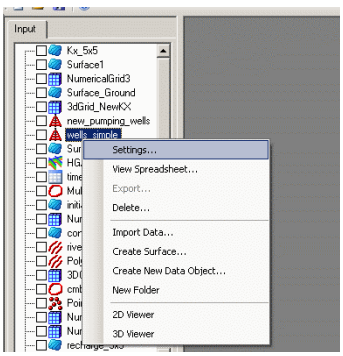
Data Settings and Properties

Hydro GeoBuilder allows you to view and modify various settings and properties for each imported or digitized data object. In general, data object settings consist of the following categories:

- **General**: View data object metadata including source data information, statistics, native coordinate system, field mappings; view the attribute and geometry table.
- **Operations**: Apply arithmetic and geometric operations to data object geometry and attributes.
- **Style Settings**: Modify various style settings for different data object elements, *e.g.*, point/line symbology, show labels, color rendering by attribute, create isosurfaces, set transparency etc.

Data object settings can be accessed from the main Hydro GeoBuilder window. To view the settings for a data object, follow the steps below:

- In the **Data Explorer**, right-click on the desired data object and select **Settings...** from the pop-up menu.



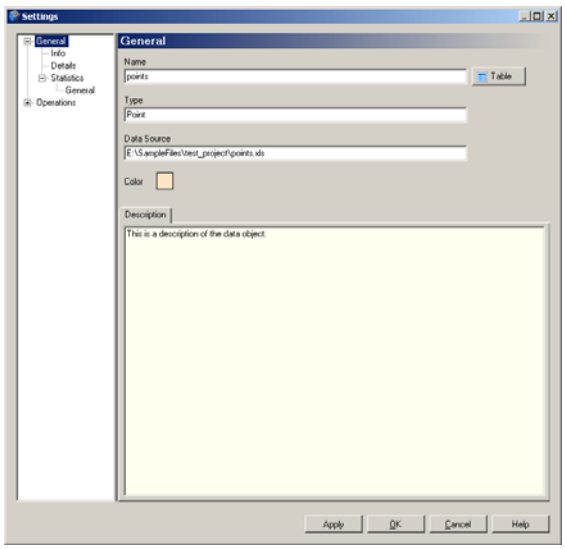
This chapter presents information on the following topics:

- Viewing General Data Settings
- Viewing the Data Table
- Performing Operations on Data

- Modifying Data Style Settings

4.1 Viewing General Data Settings

General settings consist of data object meta data including the coordinate system, field mappings, and source data information. General settings can be accessed by expanding the **General** node in the **Settings** tree. The settings in each sub-node are described below.



Info

The **Info** node contains the following information:

Name: The name of the data object (read only) defined during import.

Type: The type of the data object, *e.g.*, points, polygon, polyline.

Data Source: The folder path of the data source when the data was imported.

Color: The color of the data object when displayed in 2D and 3D Viewers. Click the color box to select a new color for the data object.

Description: The description of the data object as defined during import. This field is editable allowing you to update the description at any time.

Table (Button): The table button allows you to view the geometry and attributes of the data object in a table view. For more information on the table view, please see “Viewing the Data Table” on page 63.

Details

The **Details** node contains information on the coordinate system of the data object, and the field mappings specified during import.

Statistics

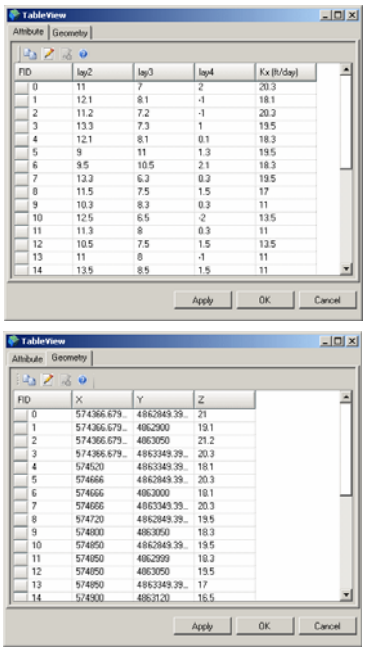
The **Statistics** node displays a count summary of the various data elements that comprise the data object. For example, for polygon data objects, the statistics node will show the number of polygons, polygon parts and vertices in the data object. For cross section data objects, the number of wells and cross sections is displayed.

For **Property** and **Structural Zone** conceptual model objects, various statistics are automatically calculated and displayed under the statistics nodes, including: **Min** and **Max X**, **Y** and **Z** values, **Area** and **Volume**.

4.2 Viewing the Data Table

4.2.1 General

The data table allows you to view the geometry and attribute values of a data object. The data table can be accessed from the **General** settings in the **Settings** dialog, or it can be launched by right-clicking on the data object in the **Data Explorer**, and selecting, **Spreadsheet...** . **Note: Map** and **Cross Section** data objects do not have a data table.



The **Attribute** tab contains the attribute data of the selected data object. Each column in the attribute tab represents an attribute. The **Geometry** tab contains the geometry (X,Y, Z) values of the data object. In both tabs, the **FID** column uniquely identifies each feature in the data object.

To copy data to the Windows clipboard, highlight the data to be copied, and then click the **Copy** button, or press **CTRL+C** on your keyboard.

Tip! When a data object is shown in 2D Viewer and the viewer is set to **Pick Mode**, you can select a row from the attribute or geometry table, and the corresponding feature will be highlighted in the 2D Viewer.

Modifying Attribute and Geometry Data

Imported geometry and attribute data can be modified in the **Data Table**. To make changes to data, click the **Begin Edit** button to enter edit mode. Make the necessary changes to the data table and once finished, click the **End Edit** button to save the changes. Please note that the data table only allows you to modify existing attribute and geometry data. Currently, Hydro GeoBuilder does not allow you to you to create new columns, *i.e.*, new data object attributes. This can only be done during the data import process.

4.2.2 Well Table

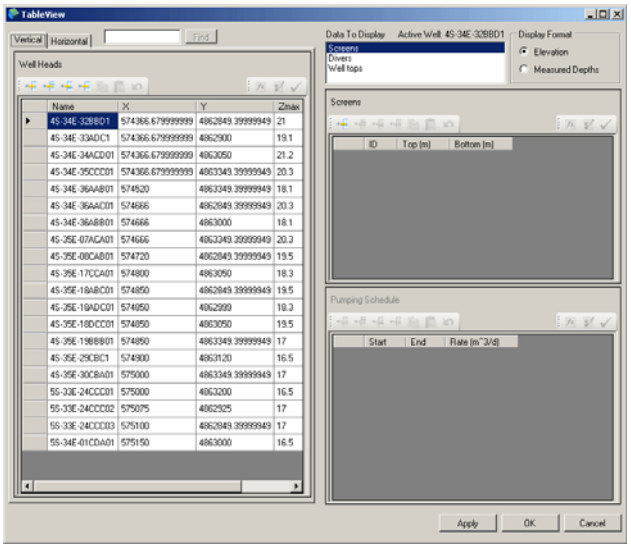
For **Wells** data objects, the data table is different than that of other data objects. The **Well** data table is designed to allow you to add and/or modify wells and associated well data, *e.g.*, pumping schedule, screen intervals, observation points, etc.

To access the well table, right-click on a **Wells** data object in the **Data Explorer**, and select **Settings...** . In the **Settings** dialog, click on the **Table** button.

In the **Well Table,** there are two tabs: **Vertical** and **Horizontal**. Each tab is described in the following sections.

Vertical Wells

The **Vertical** tab allows you to view and modify data for vertical wells.



The **Well Heads** table contains a list of all the wells in the data object. The data stored in this table includes the **Name**, **X-Y coordinates**, **Elevation** and **Depth** for each well. When a well is selected, its corresponding attribute data is displayed in the adjacent data tables, *e.g.*, Screens, Pumping Schedule. You can **search** for a well in the **Well Heads** table by entering the well name in the text box, located at the top of the window, and then clicking the **[Find]** button.

The **Data to Display** list box allows you to select which tables to display. For example, if **Screens** is selected (default), the **Screens** and **Pumping Schedule** tables will be shown. If **Divers** is selected, the **Observation Points** and **Observation Data** tables are shown. If **Well Tops** is selected, the **Well Tops** table will be shown.

The **Display Format** frame allows you express the Z values in the data tables as either an **Elevation** or a **Measured Depth** (with respect to the well head **Zmax**).

At the top of each table, there is a set of buttons that allow you to add, remove and modify the contents. These buttons are described below:

Add a row to the table

Insert a row **above** the active row

Insert a row **below** the active row

Remove the active row from the table

Adding Well Head Data

To add an item to the **Well Head** table, follow the steps below:

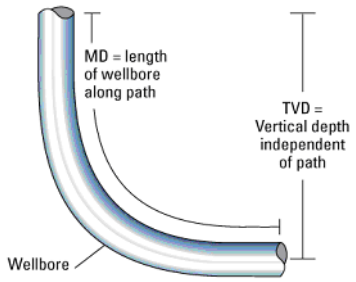
- Click the **Add Row** button from the Well Head toolbar to add a new item to the table.
- Enter an alphanumeric name in the **Well Name** column. **Note:** The well name must be unique and it may contain hyphens and spaces, but not the forward or backward slash characters.
- Enter the **X-Y** coordinates of the well head in the **X** and **Y** fields, respectively.
- Enter the elevation of the well head in the **Zmax** field.
- Enter the depth of the well in the **Zmin** field.

Adding Well Screens

To add an item to the **Well Screen** table, follow the steps below:

- Make sure the **Screens** option is selected from the **Data to Display** box.
- Select a **Well** from the **Well Head** table
- Click the **Add Row** button from the **Screens** toolbar to add a new item to the table.
- Enter a screen identification number in the **Screen ID** field.
- Enter a screen top elevation (or measured depth) in the **Screen Top** field.
- Enter a screen bottom elevation (or measured depth) in the **Screen Bottom** field.

Note: For **Horizontal Wells**, the screen values for **Screen Top** and **Screen Bottom** should be entered as a measured depth (MD) along the wellbore, with respect to the well head (see image below).



Adding Pumping Well Schedule

The **Pumping Schedule** table is used to enter the well pumping rates for specified time periods. Negative pumping rate values are used for extraction wells, and positive pumping rates are used for injection wells.

Pumping well schedules are defined for well screens, and therefore a screen must exist before a pumping schedule can be defined.

To add pumping schedule items to the pumping schedule table, follow the steps below:

- Make sure the **Screens** option is selected from the **Data to Display** box.
- Select a **Well** from the **Well Head** table, and a **Screen** from the **Screen Table** (if multiple screens exist)
- Click the **Add Row** button from the **Pumping Schedule** toolbar to add a new item to the table.
- Enter a **Start** time value and press the **<Tab>** key to advance to the **End** time field.
- Enter an **End** time value and press the **<Tab>** key to advance to the **Rate** field.
- Enter a pumping Rate value (remember to use a negative value for extraction wells)
- Press the **<Tab>** key again to create a new schedule item.
- The final time in the pumping schedule should have a pumping rate of 0 to indicate the stop time.

Note: If the pumping schedule is not specified for the entire length of the transient simulation, then it will assume the well is shut off for the time where no information is available. For steady-state simulation, the pumping rate for the first time period will be used as the steady-state pumping rate.

Adding Observation Points

Observation Points are the elevations at which head or concentration observations are recorded. Although most monitoring or observation wells are installed with a well screen spanning a known interval of the aquifer, Hydro GeoBuilder requires a single observation point elevation to be defined instead of a well screen interval. For more information on how Visual MODFLOW handles observation points please refer to the "Observation Points" in the Visual MODFLOW User's Manual.

To add an observation point for a well, follow the steps below:

- Select the **Divers** option from the **Data to Display** box. This will show the **Observation Points** table.
- Select a well from the **Well Heads** table
- Click the **Add Row** button from the **Observation Points** toolbar to add a new item to the table.
- Enter an observation point ID in the **ID** field.
- Enter an elevation value in the **Elevation** field.

The MODFLOW simulator supports head and concentration observation wells with multiple observation points throughout the length of the well-bore. Repeat the steps above to add additional points.

Adding Observations

The **Observation Data** table is used to enter the observed values at specified times, for the selected observation point.

To add observations to the Observation Data table, follow the steps below:

- Select **Divers** from the **Data to Display** box.
- Select the well from the **Well Heads** table for which observation data will be added.
- Select the desired observation point from the **Observation Points** table.
- Click the **Add Row** button from the **Observation Data** toolbar to add a new item to the table.
- Enter the time at which the head or concentration was observed in the **Time** field.
- Enter the observed head or concentration in the **Head** and **Conc1**, **Conc2**, etc., fields, respectively.

Adding Well Tops

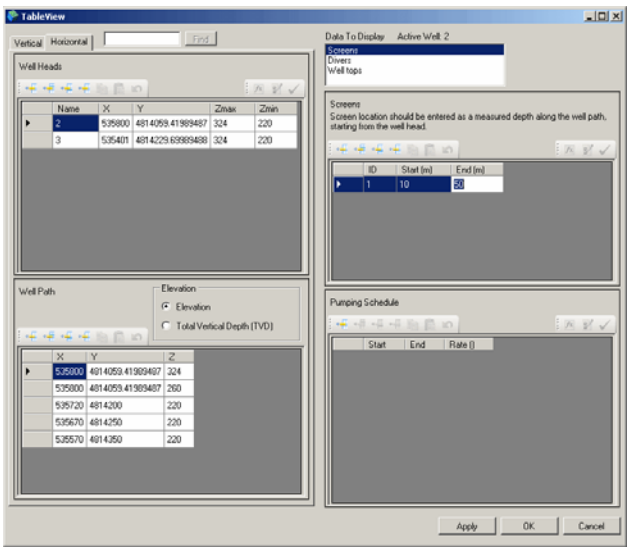
The Well tops table is used to enter the elevation points along the well path, where the well intersects with a horizon. Well top information can be used in Hydro GeoBuilder to create surfaces which can then be used to define conceptual model horizons. For more information on creating surfaces from well tops, please see "Converting Well Tops to Points Data Object" on page 74.

To add well top information to the Well Top table, follow the steps below:

- Select Well Tops from the **Data to Display** box.
- Select the well from the Well Heads table for which well top data will be added.
- Click the **Add Row** button from the **Well Tops** toolbar to add a new item to the table.
- Enter the location of the well top as a measured depth in the **Depth** field.
- Enter the name of the formation, *e.g.*, Clay, Sand etc., in the **Formation** field.

Horizontal Wells

The **Horizontal Well** tab is used to store information on deviated (horizontal wells) and contains many of the features available in the Vertical Well tab, *e g.*, add/modify screens, pumping schedule observation points etc. Please refer to the previous section for a description of these common features. The primary difference of the horizontal well tab is the ability to view and modify the deviated **Well Paths**.



Well Path Table

The **Well Path** table is used to view, add and modify the well paths for horizontal wells. When a well is selected from the **Well Heads** table, its corresponding well path geometry data is displayed in the **Well Path** table.

Horizontal well paths consist of a series of points with known coordinates and elevation that represent nodes along the well path trajectory. Hydro GeoBuilder then creates the well path by connecting each node in the series.

The **Elevation** frame allows you specify the Z value of the well path nodes as either an **Elevation** or as a **Total Vertical Depth** (measured from ground surface).

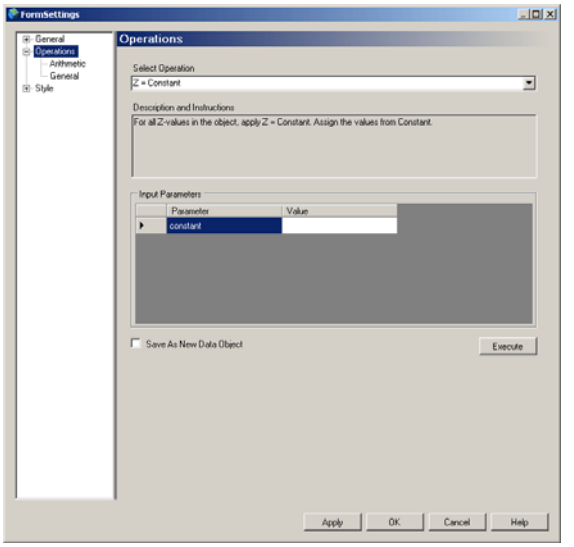
Adding Well Path

To add a new well path node to the **Well Path** table, follow the steps below:

- Select the Well from the **Well Heads** table for which the well path will be created.
- Click the **Add Row** button from the **Well Path** toolbar to add a new item to the table.
- Enter the **X**, **Y** and **Z** value of the well path node in the table. If this is the first row in the table, you would enter the **X**, **Y** and **Z** values of the well head, *i.e.*, the top-most node in the well path.
- Repeat above for additional nodes.

4.3 Performing Operations on Data

For most data object types, Hydro GeoBuilder allows you apply various arithmetic operations to your source data. Operation settings can be accessed by clicking on the **Operations** tree node in the **Settings** window. (To access the **Settings** window, in the **Data Explorer**, right-click on the data object and select **Settings...** from the pop-up menu).



Hydro GeoBuilder supports the following data operations:

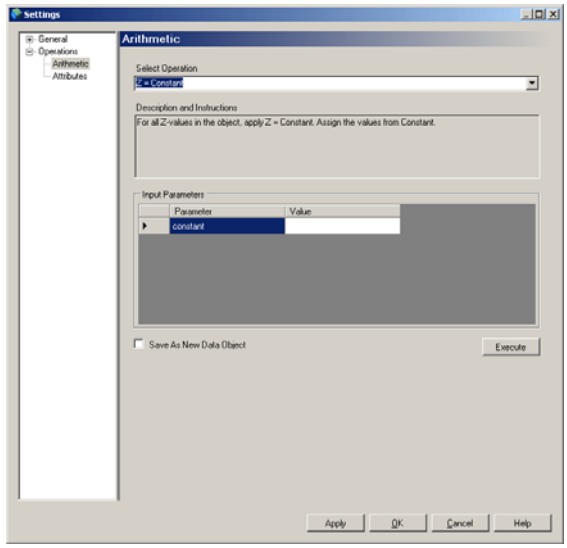
- **Arithmetic Operations** *Polygons, Polylines, Points and Maps only*
- **Attribute Operations** *Surface, Points, Time Schedules only*
- **Converting Model Layers to Points Data Object** *Cross Sections only*
- **Converting Well Tops to Points Data Object** *Wells only*
- **Calculating Well Head Elevation (Z) from a Surface** *Wells only*

Each type of operation is described in detail in the following sections.

4.3.1 Arithmetic Operations

Arithmetic operations allow you to shift the *elevation* values in the data object source data according to a user specified arithmetic expression. For example, you can use this option to drape a map over a specified surface data object. You can also shift polylines/ polygons/points up or down by a specified constant value. Arithmetic operations can be applied to the following data objects: **Polygons**, **Polylines**, **Points** and **Maps** only.

When the **Arithmetic** node is selected from the **Settings** tree, the following dialog will display:



To apply an arithmetic operation follow the steps below:

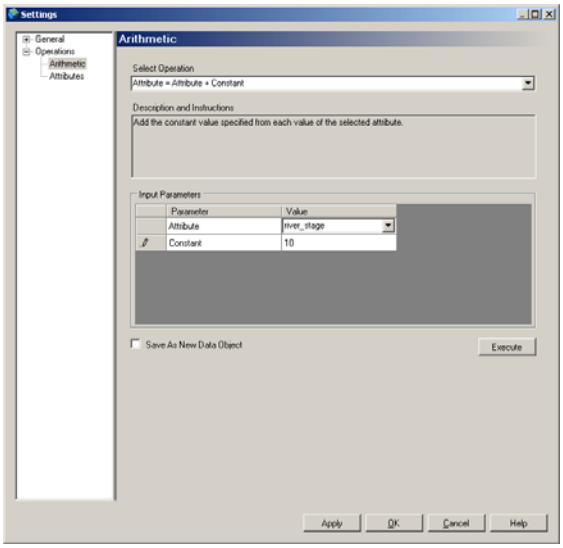
- Select the desired arithmetic expression from the **Select Operation** combo box.
- If you are unsure of what the expression does, refer to the provided description in the **Description and Instructions** text box.
- The contents of the **Input Parameters** frame will vary depending on the selected expression.
- If the selected expression contains a **constant** value, *e.g.*, Z = Constant, enter a value in **Value** field.
- If the selected expression requires a **surface**, *e.g.*, Z = Surface(x,y), then select the desired surface from the **Data Explorer**, and then click the button to insert the surface into the **Value** field.
- **Optional**: Select the **Save As New Data Object** check box to save the transformed data as a new data object.
- Click the **[Execute]** button to apply the operation.

Note: If the data object is being viewed in a 3D Viewer while the operation is applied, you may have to turn off the data object, and then turn it back on to see the changes.

4.3.2 Attribute Operations

Modify an Attribute using a Constant Value

For time-schedule data objects, Hydro GeoBuilder allows you to modify attribute values using a specified constant value. For example, the constant value can be set equal to, added to, subtracted from, and multiplied by the existing attribute values.



To apply an attribute operation follow the steps below:

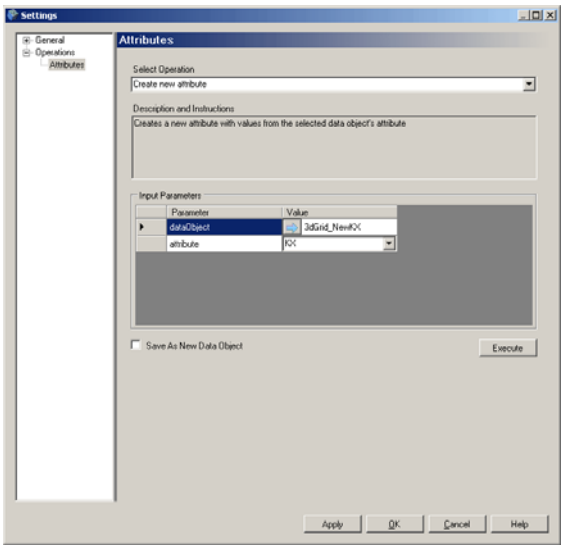
- Select the desired expression from the **Select Operation** combo box.
- If you are unsure of what the expression does, refer to the provided description in the **Description and Instructions** text box.
- In the **Input Parameters** frame, select an attribute from the combo box under the **Value** column.
- Enter a value in the **Constant** field, under the **Value** column.
- **Optional**: Select the **Save As New Data Object** check box to save the transformed data as a new data object.

Click the **[Execute]** button to apply the operation

Creating an Attribute from 3D Gridded Data Object

For Points and Surface data objects only

This operation allows you to create a new attribute using 3D Gridded data for surface and points data objects. This feature can be useful after you have run the numerical model simulation using Visual MODFLOW, and you have imported the .HDS file back into Hydro GeoBuilder as a 3D Gridded data object for visualizing the heads in 3D Viewer. The head information in the 3D Gridded data object can be extracted, and interpolated for a surface or points data object. You can then use the **Color by Attribute** feature (See “Color By Attribute” on page 82) to display the heads information on the surface or points data object. Likewise, this can be used for visualizing any attribute contained in a 3D Gridded data object on a surface or points data object. This procedure is described below.



This operation can be accessed from the **Settings** dialog. Select the surface or points data object in the **Data Explorer**, right-click and then select **Settings...** . Once the **Settings** dialog launches, expand the **Operations** node, and select **Attribute** from the settings tree.

- Select **Create new Attribute** from the **Select Operation** combo box
- Select the **3D Gridded** data object from the **Data Explorer**, and select the button to insert the data object into the **dataObject** field.
- Once the 3D Gridded data object is selected, its available attributes are populated in the **Attribute** combo box. Select the desired attribute from the Combo box.
- **Optional**: Select the **Save As New Data Object** check box to save the transformed data as a new data object.
- Click the **[Execute]** button to apply the operation.

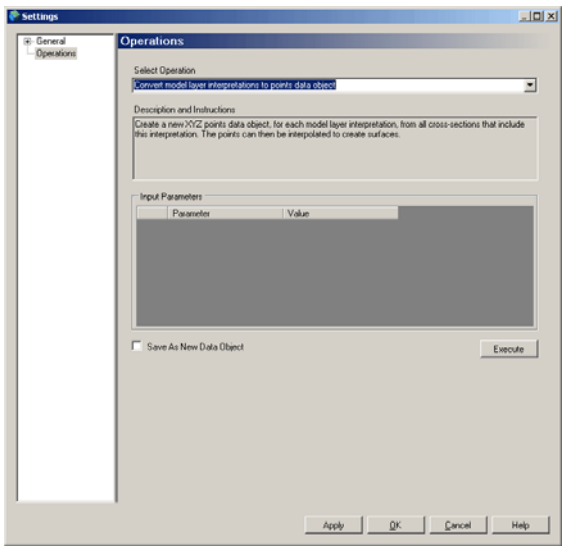
Once the operation is applied, you can confirm that the new attribute was created by viewing the table view for the selected surface or points data object.

4.3.3 Converting Model Layers to Points Data Object

For Cross Section data objects only

This operation allows you to create a new points data object, for each model layer interpretation, from all cross sections in the data object that include this interpretation. Once the points data objects are created, you can then create surface data objects, which can then be used to define the horizons of your conceptual model.

Note: This feature is available for model layer interpretations only.



To create points data objects from cross section interpretation model layers, follow the steps below:

- From the **Select Operation** combo box, select **Convert Model Interpretations to Points Data Object** (default).
- Click the **[Execute]** button to apply the operation.

Once the points data objects are created, they will be added to the **Data Explorer**, where they can be used to create surface layers.

4.3.4 Converting Well Tops to Points Data Object

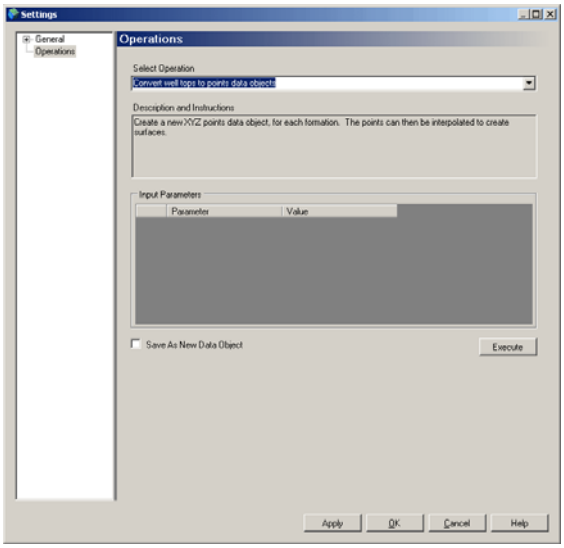
For Wells data objects only

This operation allows you to create a new points data object, for each well top formation in a wells data object. The resulting points data objects can then be used to create

surfaces, which can be used to define the horizons of a conceptual model.

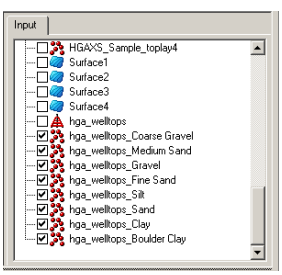
Well top data can either be included during data import, or they can be manually defined in the well table view. For information on defining well tops, please see “Adding Well Tops” on page 68.

To access this operation, right-click on the desired wells data object in the **Data Explorer**, and select **Settings...** from the pop-up menu. In the Settings dialog, select the **Operations** node, and a window, similar to the one shown below, will display.



To create new points data objects from well top formation, follow the steps below:

- Select the **Convert well tops to points data objects** option from the **Select Option** combo box (selected by default).
- Click the **[Execute]** button to apply the operation.

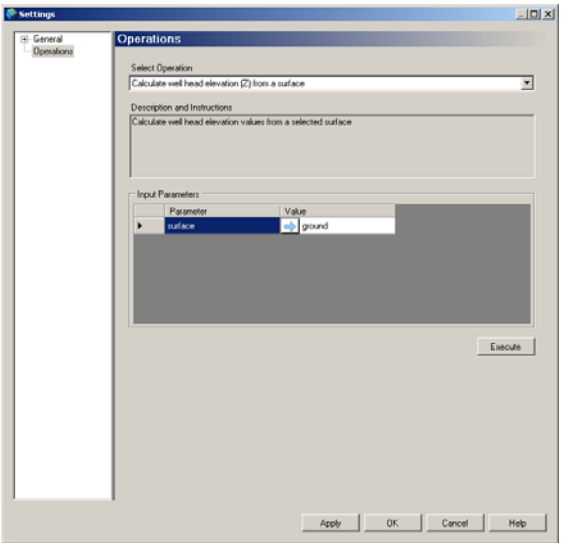


Once the points data objects are created, they are added to the **Data Explorer** using the naming convention [*wells data object name*]_[*formation label*] (shown above).

4.3.5 Calculating Well Head Elevation (Z) from a Surface

For well data objects only

This operation is only available for well data objects. It allows you to calculate elevation values for each well head in the data object, using a specified surface data object. Please note, any well head elevations that have been added manually or imported will be overwritten with the elevation values calculated from the specified surface.



To calculate well head elevation from a surface,

- Select **Calculate well head elevation (Z) from a surface** from the **Select Operation** combo box
- Select the desired **Surface** data object from the **Data Explorer**, and select the button to insert the data object into the **surface** field.
- Click the **[Execute]** button to apply the operation.

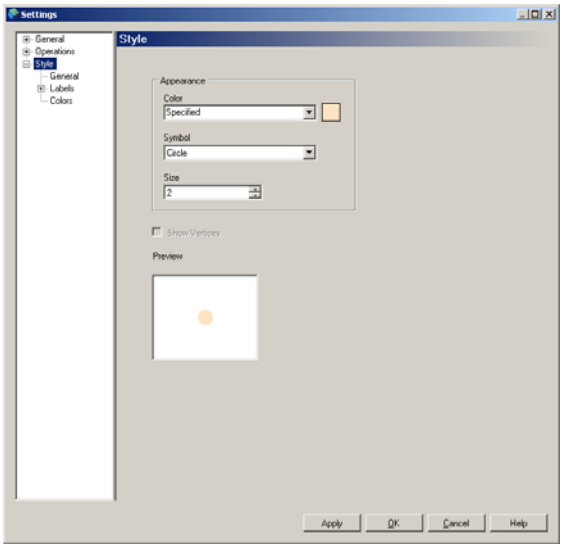
Once the operation is applied, you can confirm that the new Z values were created by viewing the table view for the selected well data object. Please note that Hydro GeoBuilder will ignore wells where the elevation of the bottom of the well (Zmin) is greater than the calculated well head elevation.

4.4 Modifying Data Style Settings

Hydro GeoBuilder provides you with a wide-variety of style settings, allowing you to modify the appearance of data objects in both 3D and 2D Viewer. The style settings can be accessed by expanding the **Style** node in the **Settings** tree (shown below). Please

note that the **Style** node will only be available when the particular data object is being shown in 2D or 3D Viewer.

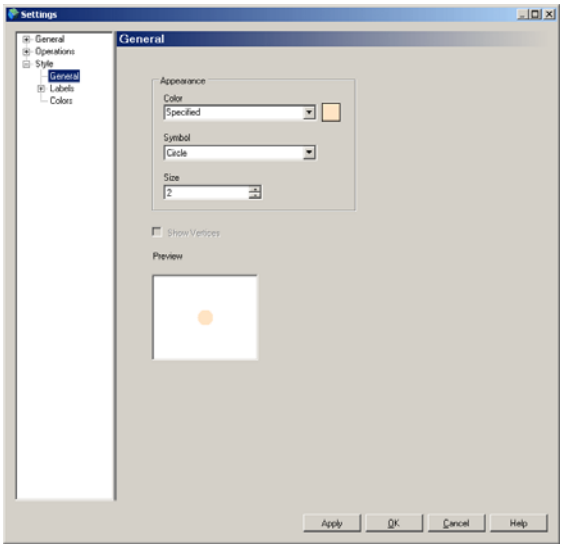
In general, each data object has its own set of style settings, although some settings are common between data objects. The following sections describe the style settings for the various data objects.



4.4.1 Points, Polygons & Polylines

Points \ Vertices

The style settings for points data objects are described below:



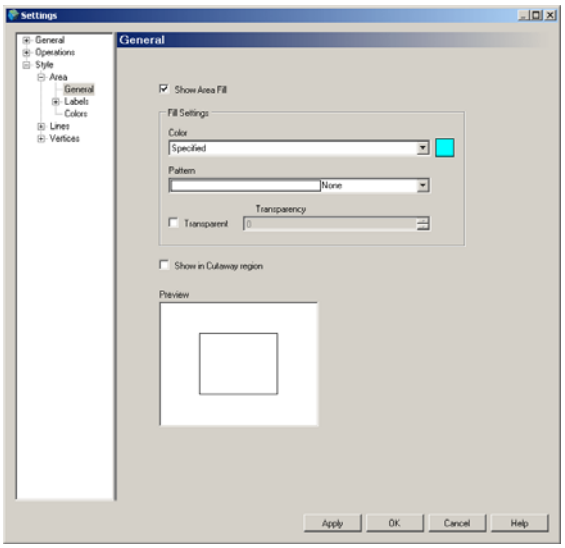
Select the **color** method, **symbol**, and **size** from the appropriate combo boxes.

Choose from the following color methods:

- **Specified**: Points are colored using the color specified in the **General** settings.
- **By Attribute:** Points are colored based on a specified data object attribute. See “Color By Attribute” on page 82 page for more information on color rendering.

The **Show in Cutaway** check box allows you to show points or vertices in cutaway regions in the 3D Viewer window. When this option is disabled, points or vertices will not show in any areas that have been “hidden” in the 3D Viewer window by creating “Cutaways”. For information on creating cutaways, please see “Creating Cutaways” on page 101.

Area



Select the **Show Area Fill** check box to show/hide the area file. If unchecked, only the shape boundary will be visible in 2D and 3D Viewers.

Select the **color** method and fill **pattern** symbology, from the appropriate combo boxes.

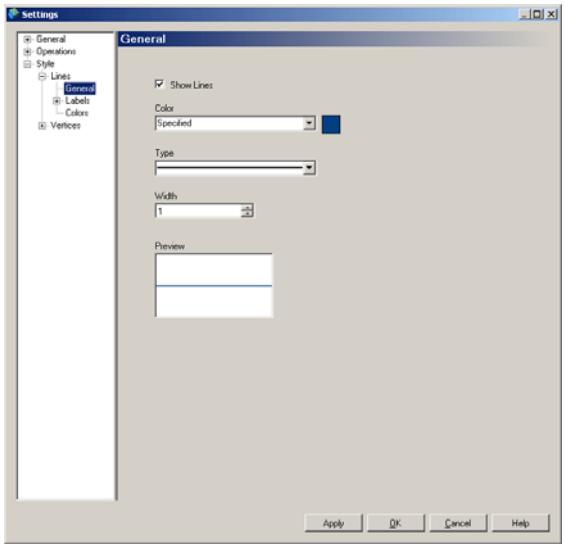
Choose from the following color methods:

- **Specified**: Shape is colored using the color specified in the General node
- **By Attribute**: Shape is colored based on an attribute. See “Color By Attribute” on page 82 for more information on color rendering
- **Custom:** Specify a color for the area fill. This color will overwrite the default color defined in the general settings for this particular shape element

Select the **Transparent** checkbox to make the polygon fill pattern transparent. Use the adjacent **Transparency** text box to set the level of transparency, *e.g.*, a higher value will make the fill more transparent.

The **Show in Cutaway** check box allows you to show areas the polygon in cutaway regions in the 3D Viewer window. When this option is disabled, the polygon will not show in any areas that have been “hidden” in the 3D Viewer window by creating “Cutaways”. For information on creating cutaways, please see “Creating Cutaways” on page 101.

Lines



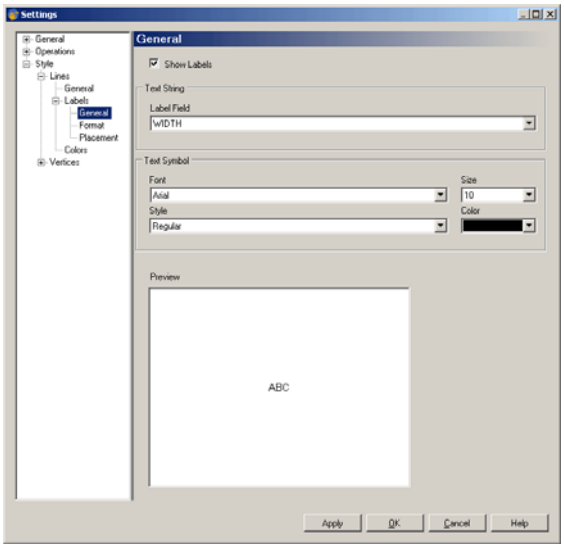
Select the **Color** method, line **Pattern** (solid or dash), line **Width** from the appropriate combo boxes.

Choose from the following color methods:

- **Specified**: Line is colored using the color specified in the **General** settings.
- **Custom:** Specify a color for the line element. This color will overwrite the default color defined in the general settings for this particular shape element

The **Show in Cutaway** check box allows you to show areas the line in cutaway regions in the 3D Viewer window. When this option is disabled, lines will not show in any areas that have been “hidden” in the 3D Viewer window by creating “Cutaways”. For information on creating cutaways, please see “Creating Cutaways” on page 101.

Labels



General

The following label settings are available under the **General** node (shown above):

Show Labels: Show/Hide the labels in 2D/3D viewer.

Label Field: This combo box contains all attributes for the selected data object. Select the desired attribute field to use for the labels.

Font: Select the label font from the combo box.

Size: Set the text size of the labels.

Style: Set the font style for the labels, *e.g.*, Regular, Bold, Font, Italicized, etc.

Color: Set the color of the label text.

Format

The following label settings are available under the **Format** node:

Format: Choose between **Numeric** or **Scientific** notation

Decimals: Set the number of decimals to plot for each label.

Placement (Polylines Only)

Position: Display the label **above**, **below**, or **on the line**.

Offset: This parameter controls how far the label will be placed from the line.

Interval: To be updated.

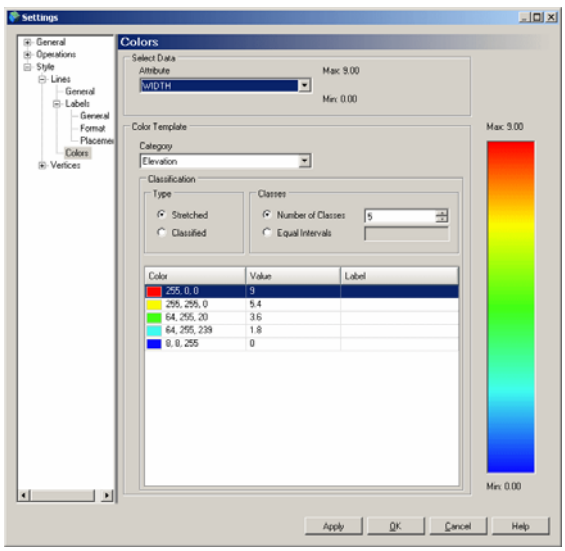
Location along the line: Display the label at the **start**, in the **middle** or at the end of the line.

Orientation to the line: Display the label **parallel**, **perpendicular** or **horizontal** to the line.

Color By Attribute

Data objects can be colored based on a specified attribute. Color rendering can be applied to any shape element that contains attributes. To color a data object by attribute, follow the steps below:

- In the properties dialog, expand the **Style** node and select the shape element to be colored, *e.g.*, line.
- From the **Color** combo box, select the **ByAttribute** option
- From the **Settings** tree, select the **Colors** node.



- Select the desired attribute from the **Attribute** combo box. You will notice that the min and max values are displayed to the right of the combo box.
- Select the color template from the **Category** combo box. **Note**: Currently, Hydro GeoBuilder only supports the “Elevation” color template. In future releases additional color templates will be available for you to choose from. For the **Elevation** color template, you can define various settings. These settings are described below.

Type: Select between **Stretched** or **Classified**. The stretched option allows for color shading, *i.e*, continuous color gradient. The classified option allows for discrete intervals, *i.e.*, zebra, in the color gradient.

Classes: There are two options for defining the number of classes to be used: **Number of Classes** and **Equal Intervals**.

Number of Classes: Specify the number of color classes in the text box.

Equal Intervals: Specify an interval spacing, and the number of classes will be determined from the range of the attribute data. For example if your data rage is 100, and you specify a 10 interval spacing, 10 classes will be created.

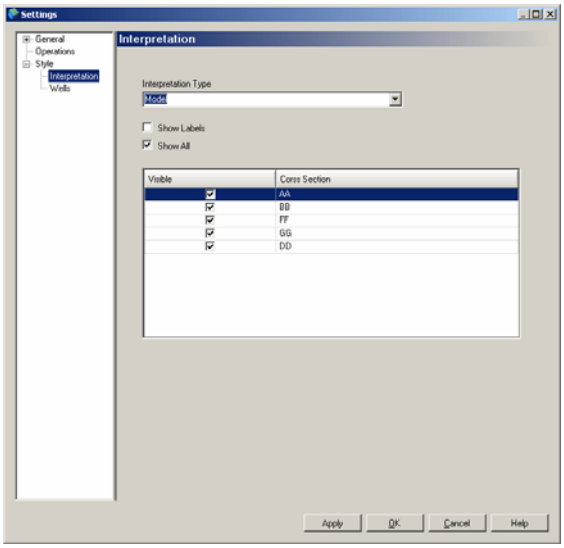
As the number of intervals/classes is modified, rows are automatically removed or added to the color table. For the **Elevation** color scheme, the maximum value will always be **Red** and the minimum value will always be **Blue**, but the gradient between will change automatically depending on the specified number of classes.

- Hydro GeoBuilder automatically calculates the intervals based on the defined specified number of classes or equal intervals. However, if desired, you can manually edit the intervals in the grid under the **Value** column.
- Once the settings have been defined, click the **[Apply]** button to show the changes in an active 2D or 3D Viewer.

4.4.2 Cross Sections

The following section describes the available style settings for cross section data objects.

To access the style settings, right-click on the cross section data object in the **Data Explorer**, and select **Settings...** from the pop-up menu. Then, in the **Settings** dialog, expand the **Style** node to view the style settings.



Cross section data objects consist of two main elements; the interpretation layers and the cross section wells. The settings for each element can be accessed by clicking on the **Interpretation** or **Wells** node, respectively.

Interpretation

A screen capture of the interpretation settings is shown above.

From the **Interpretation Type** combo box, select which interpretation layer to show in 3D Viewer. Select from **Model**, **Geology** or **Hydrogeology**.

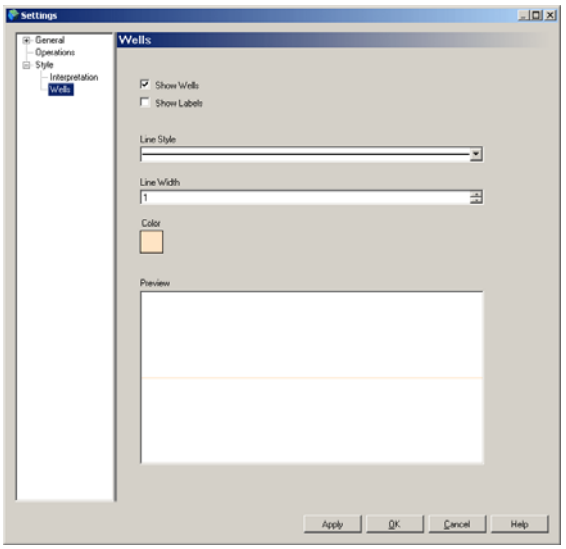
When a interpretation layer is selected from the combo box, its associated cross sections are listed in the grid below. Under the **Visible** column, select which cross section to show/hide in 3D Viewer.

Select the **Show All** check box to show all the cross sections for the selected interpretation layer.

Select the **Show Labels** check box to show the label for each cross-section.

Wells

The wells node contains settings for changing the appearance of the cross section wells. These settings are described below.



Show Wells: Check this option to show the well geometry.

Show Labels: Check this option to show the well label above each well.

Line Style: Select the type of line to display. Choose between solid or dashed.

Line Width: Specify the width of the wells.

Color: Change the color of the wells.

Click the **[Apply]** button to display the changes in an active 3D Viewer window.

4.4.3 Wells

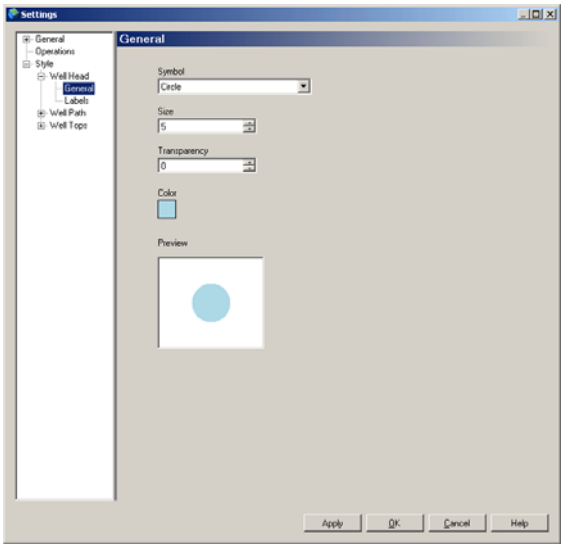
The following section describes the available style settings for **Wells** data objects.

To access the style settings, right-click on the well data object in the **Data Explorer**, and select **Settings...** from the pop-up menu. Then, in the **Settings** dialog, expand the **Style** node to view the style settings.

The Wells settings are divided into three sections: **Well Head**, **Well Path** and **Well Tops**.

Well Head

The **Well Head** node provides options for changing the appearance of the well heads (top of the wells).



When the **General** subnode is selected, the following options are available:

Symbol: Select the well head symbology from the combo box. Choose from various symbols including circle, square, cross, diamond etc.

Size: Specify the size of the well heads.

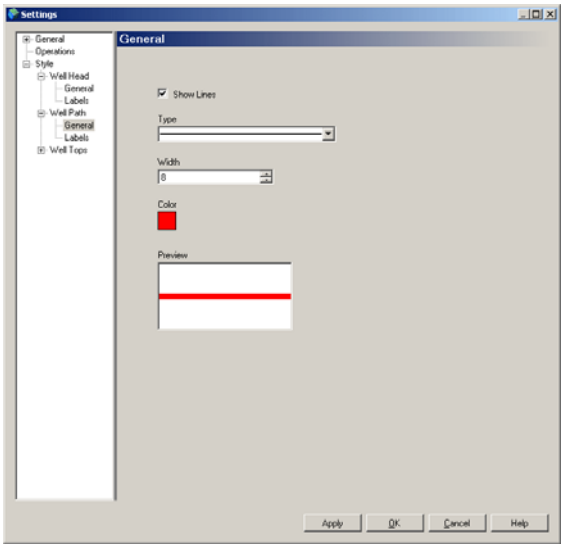
Color: Select a color the well heads.

Click the **[Apply]** button to view the change in an active 2D or 3D Viewer.

For information on the **Labels** node, please see “Labels” on page 81.

Well Path

The **Well Head** node provides options for changing the appearance of the well path. These options are described below.



When the **General** subnode is selected, the following options are available:

Show Lines: Select this option to show/hide the well path.

Type: Specify the line type for the well path. Choose between a solid line or a dashed line.

Width: Specify the width of the well paths.

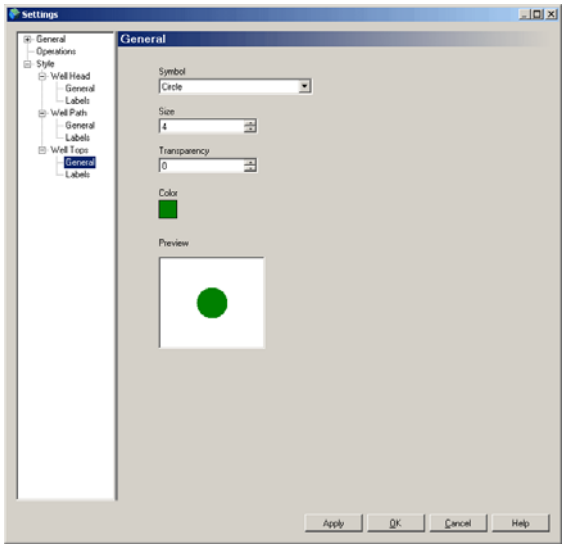
Color : Select a color the well paths.

Click the **[Apply]** button to view the change in an active 2D or 3D Window.

For information on the Labels node, please see “Labels” on page 81.

Well Tops

The **Well Tops** node provides options for changing the appearance of the well tops. These options are described below.



When the **General** subnode is selected, the following options are available:

Symbol: Select the well top symbology from the combo box. Choose from various symbols including circle, square, cross, diamond etc.

Size: Specify the size of the well top symbols.

Color: Select a color for the well tops.

Click the **[Apply]** button to view the change in an active 2D- or 3D-Window.

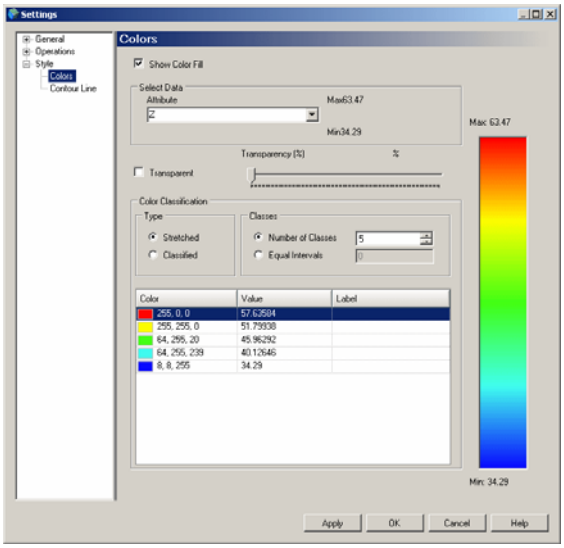
For information on the Labels node, please see section “Labels” on page 81.

4.4.4 Surfaces

The following section describes the available style settings for **Surface** data objects.

To access the style settings, right-click on the surface data object in the **Data Explorer**, and select **Settings...** from the pop-up menu. Then, in the **Settings** dialog, expand the **Style** node to view the style settings.

Colors



The **Colors** node provides options for coloring the surface layer by elevation value. The following options are available.

Show Color Fill: This option allows you to show or hide the color fill. If disabled, the surface will appear in the color defined in the **General** settings.

Transparency: This option allows you to make the surface appear transparent. When the **Transparent** checkbox is selected, use the adjacent slider bar to set the level of transparency.

Type: Select the type of gradient to use for coloring the surface. Select **Stretched** to use a continuous color gradient, or select **Classified** to use discrete color zones.

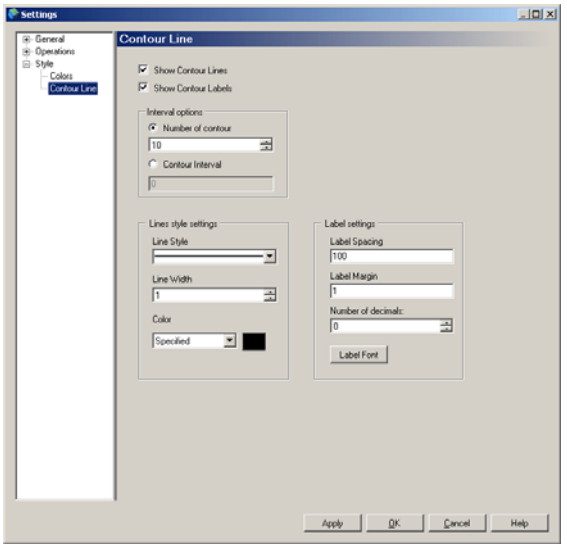
Classes: There are two options for defining the number of classes to be used: **Number of Classes** and **Equal Intervals**.

- **Number of Classes:** Specify the number of color classes in the text box.
- **Equal Intervals:** Specify an interval spacing, and the number of classes will be determined from the range of the attribute data. For example if your data rage is 100, and you specify a 10 interval spacing, 10 classes will be created.

As the number of intervals/classes is modified, rows are removed or added to the color table. The maximum value will always be **Red** and the minimum value will always be **Blue**, but the gradient between will change automatically depending on the specified number of classes.

Click the **[Apply]** button to view the changes in an active 2D- or 3D-Window.

Contour Lines



The **Contour Lines** node provides options for showing contour lines on the surface layer. The following options are available:

Show Contour Lines: Show/Hide the contour lines

Show Contour Labels: Show/Hide the contour labels

Number of Contours: Specify the number of contours to display on the surface.

Contour Interval: Set the contour increment value. When this option is used, Hydro GeoBuilder automatically calculates the number of contours used.

Line Style: Select the contour line style from the combo box. Choose between solid line or dashed line.

Line Width: Set the width (thickness) of the contour lines.

Label Spacing: Set the spacing between the contour line and the label.

Number of Decimals: Set the number of decimals to show in the contour labels.

Label Font: Select this button to specify the font settings for the contour labels.

Click the **[Apply]** button to view the changes in an active 2D or 3D Viewer.

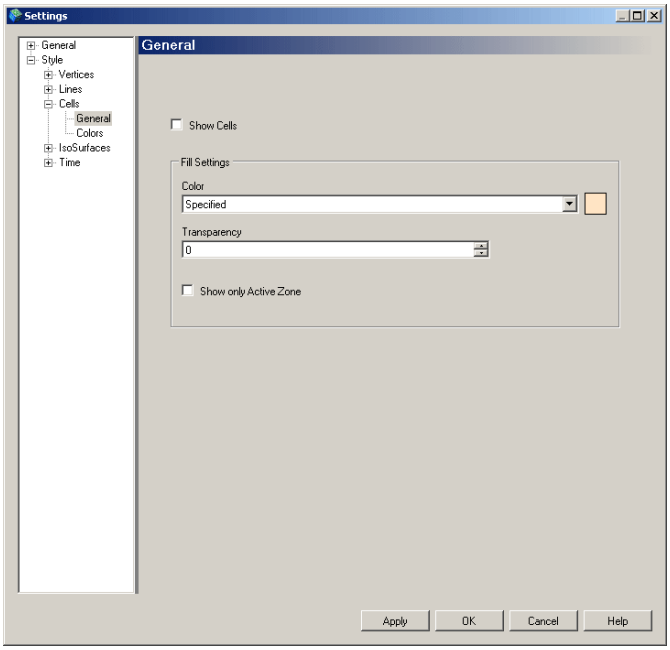
4.4.5 3D Gridded Data

The following section describes the available style settings for **3D-Gridded** data objects.

To access the style settings, right-click on the surface data object in the **Data Explorer**, and select **Settings...** from the pop-up menu. Then, in the **Settings** dialog, expand the **Style** node to view the style settings.

For information on the settings available in the **Vertices** and **Lines** nodes, please refer the “Points \ Vertices” on page 77 and “Lines” on page 80, respectively.

Cells

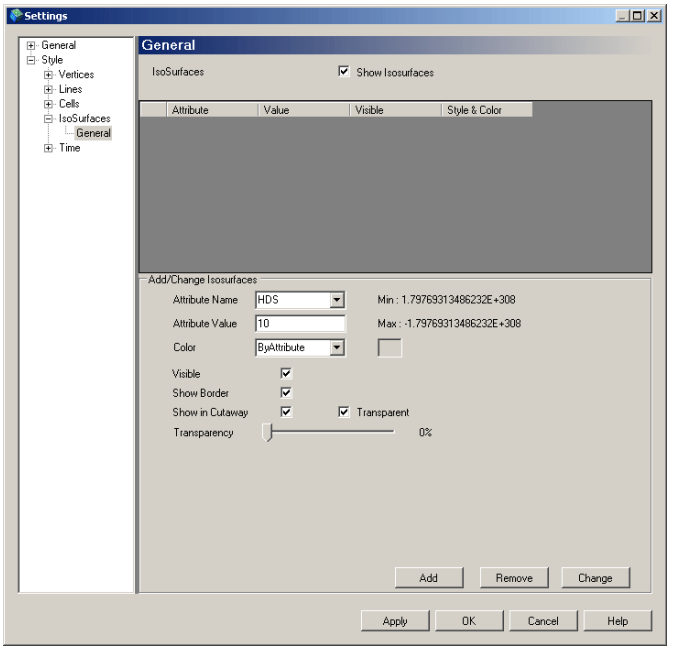


The **Cells** node allows you to specify style settings for the grid cells. The following options are available:

The **Show Cell** check box allows you to show/hide the grid cells in the 3D gridded data object. When the check box is selected, you can choose how to show the cells in the **Color** combo box in the **Fill Settings** frame. With the **Specified** option, select the adjacent color swatch and select the desired color to fill the cells. If you select **Color by Attribute**, you can color each cell according to a specified attribute, *e.g.*, heads. Color by attribute settings can be defined by selecting the **Color** node, located under the **Cells** node. For more information on the color by attribute feature, please refer to “Color By Attribute” on page 82.

The **Show only Active Zone** check box allows you to show/hide inactive grid cells.

Isosurfaces



The **IsoSurface** node allow you to create and modify one or more isosurfaces from 3D gridded attribute data. An isosurface is a 3D planar surface defined by a constant parameter value in 3D space. Isosurfaces are typically used for demonstrating the spatial distribution of a selected parameter. For groundwater modeling purposes, isosurfaces are generally used for representing the spacial distribution of heads, drawdowns and concentrations.

Creating an Isosurface

To create an isosurface, follow the steps below:

- From the **Attribute Name** combo box, select the attribute from which the isosurface is to be created.
- Specify the attribute value in the **Attribute Value** field.
- Select the color method from the **Color** box. The isosurface can be displayed as a solid color (**Custom**) or rendered by a specified attribute (**ByAttribute**).
- Use the **Visible** check box to show/hide the isosurface.
- Use the **Show Border** check box to display/hide a color map of the element value on the borders (sides) of the model domain when the isosurface intersects the edge of the model domain.
- Use the **Show in Cutaway** check box to make the isosurface visible/invisible in cutaways.
- Use the transparent check box to enable/disable transparency. If enabled, use

the **Transparency** slider to set the level of transparency/opaqueness.

- Click the [**Add]** button to create the isosurface.

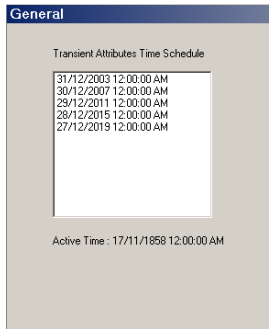
The isosurface will be added to the isosurface table.

Modifying an Isosurface

To modify an existing isosurface, follow the steps below:

- Select the isosurface from the isosurface table
- Make the modifications to the desired settings, *e.g.*, attribute name, attribute value, color, etc.
- Click the **[Change]** button to apply the changes.

Time



The **Time** node provides a list of all the time steps in the 3D gridded data object, and allows you to select the desired time step data to display in the 3D Viewer window. For 3D gridded data objects generated by steady state flow models, only one time step will be available. For 3D gridded data objects generated by transient flow models, multiple time steps will be available (as defined in the **Run** settings in Visual MODFLOW, i.e, **MODFLOW-2000 > Time Steps**).

5

Data Viewers

Hydro GeoBuilder supports two types of interactive data viewers: **3D Viewer** and **2D Viewer**. The **3D Viewer** is based on OpenGL graphics technology, allowing you to visualize graphically-rich three-dimensional representations of your data. The **2D Viewer** allows you to view your data from a planar perspective, and provides various tools for editing and drawing data objects.

This chapter presents information on the following topics:

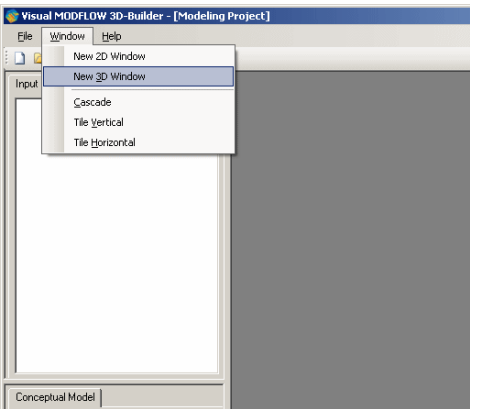
- Opening a New 2D or 3D Viewer
- Working with Viewers
- Modifying Viewer Settings
- Viewer Controls
- Linking 2D Viewers with Attribute Tables
- Exporting Viewers
- Creating a New Data Object
- Digitizing & Editing Geometry in 2D Viewers

5.1 Opening a New 2D or 3D Viewer

There are two ways in which you can launch a 2D or 3D Viewer in Hydro GeoBuilder: **From** the **Main Menu** or from the **Data Explorer**.

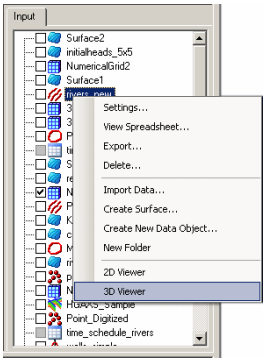
From the Main Menu

To launch a new 2D or 3D Viewer from the Main Menu, select **Window** from the Hydro GeoBuilder main menu, and select either **2D Viewer** or **3D Viewer**.



From the Data Explorer

Once data has been imported into your project, you can launch viewers from the **Data Explorer**. Right-click on a data object, and select **2D Viewer** or **3D Viewer** from the pop-up menu. A new viewer will then launch, displaying the selected data object. Please note that the viewers listed in the pop-up menu depend on which data object is selected (see table under “Displaying Data in Viewers” on page 99).

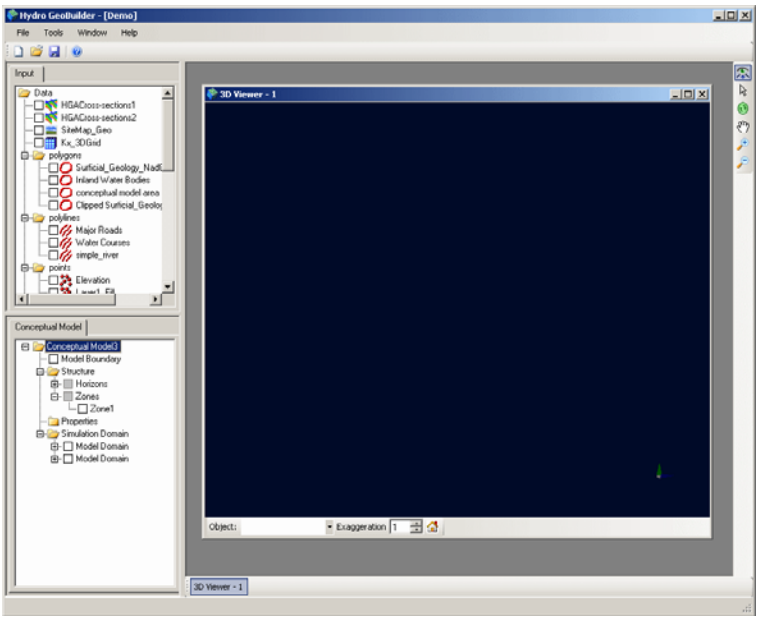


5.2 Working with Viewers

General

3D and 2D Viewers behave just like any other window. For example, you can **Minimum**, **Maximize** or **Close** the viewer by clicking the appropriate button in the top-right corner of the viewer.

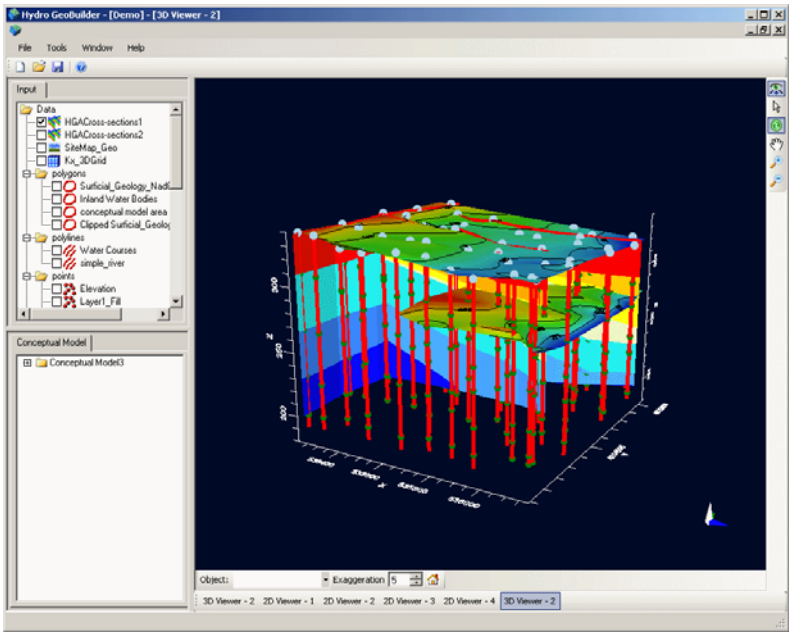
You can **Resize** the viewer by clicking and dragging the sides and corners of the viewer to a desired size, or **Move** the viewer within the Hydro GeoBuilder main window by clicking the title bar and dragging it to a new location.



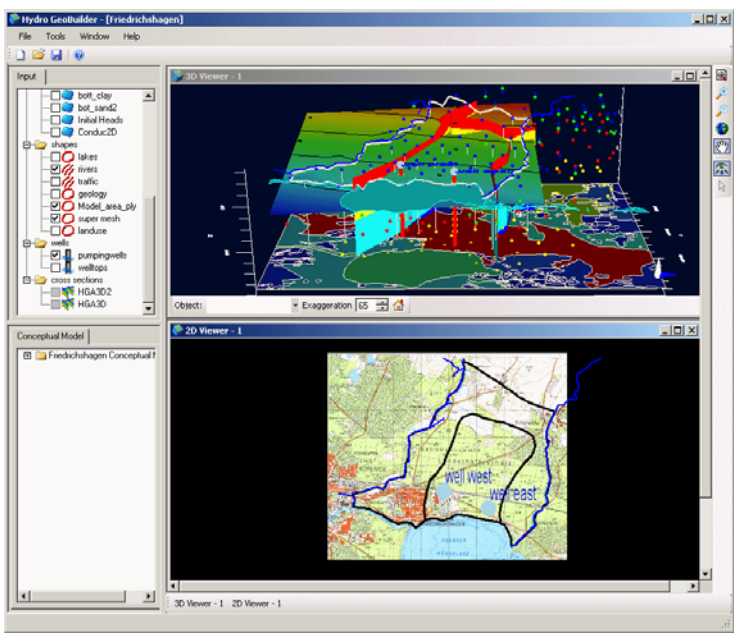
Using Multiple Viewers

Hydro GeoBuilder allows you to have multiple 2D and 3D Viewer windows opened and displayed at one time. When a new viewer is opened, it is added to the **Viewer Bar**, located at the bottom of the main Hydro GeoBuilder Window.

You can change the current active viewer by clicking on a different viewer from the viewer bar.



Viewers can be **Tiled** horizontally or vertically (shown below), or **Cascaded** within the main Hydro GeoBuilder window. These options help you organize multiple windows within the Hydro GeoBuilder window, and can be accessed by clicking on **Window** from the main menu.



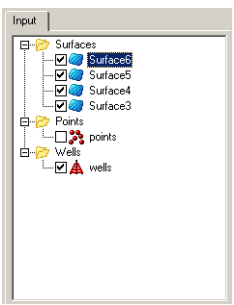
5.3 Displaying Data in Viewers

The table below summarizes which data and conceptual model objects can be displayed in the 2D and 3D Viewers.

	3D Viewer	2D Viewer
Data Objects		
Points	✔	✔
Polygon	✔	✔
Polyline	✔	✔
Surface	✔	✔
Time Schedule	N/A	N/A
Well	✔	✔
Cross Section	✔	
3D Gridded Data	✔	
Conceptual Model Objects		
Model Boundary	✔	✔
Horizons	✔	✔
Structural Zones	✔	
Property Zones	✔	
Simulation Model Domain	✔	
Boundary Conditions	✔	✔
Finite Difference Grid	✔	
Finite Element Mesh	✔	✔

General

To display data in a viewer, select the check box beside the data object in the **Data Explorer** or **Conceptual Model Explorer**. If multiple viewers are opened, the data will be shown in the active viewer.



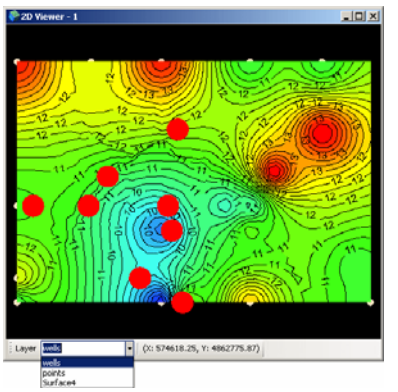
To remove data from a viewer, select the check box beside the data object so that it appears empty or “unchecked”. Please note that some data objects may not be viewable in the 3D or 2D Viewers.

When a data object is displayed in a viewer, it will appear as defined in the data object settings. For more information on viewing and modifying data object settings, please see Chapter 4: Data Settings and Properties.

Layer Ordering in 2D Viewer

Layer ordering in 2D Viewer is determined by the sequence in which data objects are added to the viewer. For example, if two data objects completely overlap each other, the data object added last will appear on top of the other.

You can bring layers to the top, by using the **Layer** combo box, located at the bottom of the 2D Viewer window.



The **Layer** combo box contains all of the layers currently displayed in the 2D Viewer. Select a layer from the combo box to bring it to “the top” of the layer order.

5.4 Modifying Viewer Settings

The following settings are available in both 3D and 2D Viewers.

Changing the Background Color

To change the background color of a 2D or 3D Viewer, right-click anywhere within the viewer, and select **Background Color** from the pop-up menu. The **Color** combo box will display on your screen. Select a new color and then click the **[OK]** button.

Showing the Viewer Axis

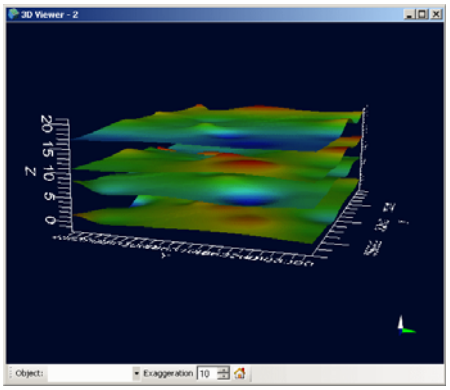
To show or hide the viewer axis, right-click anywhere within the viewer, and select **Axis** from the pop-up menu.

3D Viewer Settings

The following settings only apply to 3D Viewers.

Setting the Vertical Exaggeration

The **Vertical Exaggeration** is the ratio of the scale of the Y-axis to the scale of the X-axis. Vertical exaggeration can be used for discerning subtle topographic features or when the data covers a large horizontal distance (miles) relative to the relief (feet). By default, the vertical exaggeration is set to 0. You can change the vertical exaggeration using the **Exaggeration** text box, located at the bottom of the 3D Viewer (shown below)



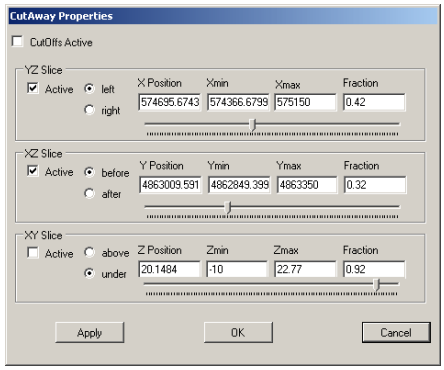
Resetting the Viewer

To zoom out to the full extents of your data click the **Reset Scene Position** button located at the bottom of the viewer. Please note that clicking this button will reset the rotation and zoom level back to the original view configuration.

Creating Cutaways

Hydro GeoBuilder allows you to remove portions of the model from the 3D Viewer by creating cutaways. To create a cutaway in a 3D-Viewer, follow the steps below:

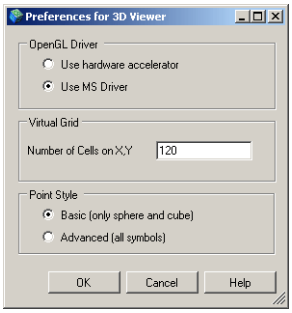
- Right-click anywhere inside the 3D Viewer, and select **CutAway Properties** from the pop-up menu. The following dialog box will display on your screen.



- Click the **CutOffs Active** check box to enable CutAways
- Under the **Activity** tab, specify which slices to make active by clicking the appropriate **Active** check boxes. The YZ slice refers to a vertical plane along the Y and Z axis, the XZ slice refer to a vertical plane along the X and Z axis, and the YZ slice refers to a horizontal plane along the Y and Z axis.
- For each active slice, select which portion to remove from the viewer (for example, the portion of the model that is left or right side of the slice, before or after the slice, or above or under the slice)
- By default, the position of each slice is automatically positioned in the middle of the data along its respective axis. However, you can manually change the position of the slice by entering a new value in the **Position** field. **Note**: The Xmin, Xmax, Ymin, Ymax, Zmin and Zmax fields are read-only, and cannot be changed.
- Alternatively, you can set the position of a slice by entering a value in the **Fraction** field. For example a fraction value of 0.5 will set the slice position in the middle of the 3D data.
- Click the **[OK]** button to apply the changes to the current 3D Viewer.

3D Viewer Performance Preferences

To access the 3D Viewer performance preferences, select **Tools / Preferences** from the main menu.



OpenGL Driver

By default, Hydro GeoBuilder will attempt to use the vendor provided driver included with your graphics acceleration hardware. If problems are encountered with the vendor provided drivers, *e.g.*, poor on-screen display/performance, then Hydro GeoBuilder provides the option to use the Microsoft Driver for OpenGL.

Virtual Grid

Depending on the size of your model, Hydro GeoBuilder may run very slowly during rotations or when data is moved in the 3D Viewer. In this situation, the virtual grid option may be used to increase the speed of the data processing and image rendering. It can be used to set up a uniformly spaced grid with a specified number of rows and columns.

The virtual grid option will interpolate the data from the model to the uniformly spaced virtual grid. This allows a smaller amount of information to be processed much faster. However, this also results in a loss of resolution of the data, and some local scale minimum and maximum values may be missed.

If you are experiencing performance issues, try lowering the **number of cells on the X,Y** axis.

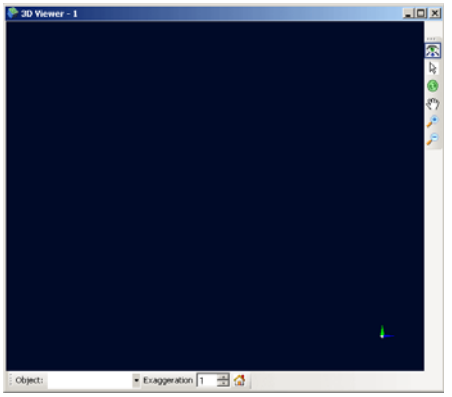
Point Style

This setting provides two options for displaying points in 3D Viewer: **Basic** and **Advanced**. If the **Basic** option is selected, 3D Viewer will render the point shapes in the 3D Viewer. On some computers this option may hinder the performance of the 3D Viewer. If the **Advanced** option is selected, 3D Viewer will use bitmap images to display the points. If you are experiencing performance issues display points in 3D Viewer, the **Advanced** option should be selected.

Note: The Basic option only supports cube and sphere symbols for displaying points.

5.5 Viewer Controls

The viewer controls allow you to interact with displayed data objects. The controls are accessible from the toolbar located along the right side of the viewer window.



The controls for both 2D and 3D Viewers are described below:

View Mode: The default mode. Allows you to zoom, rotate and pan the displayed data objects.

Pick Mode (2D Viewer Only): Allows you to select individual data object elements currently displayed in the viewer. When in **Pick Mode**, the **Edit** button will be shown in the sidebar, allowing you to edit the selected data object element. For more information on editing data objects, please see see “Digitizing & Editing Geometry in 2D Viewers” on page 107

Rotate: Allows you to rotate the displayed data objects using your mouse. Click-and-hold on the displayed data, and move the mouse in a direction to rotate the data.

Move/Pan: Allows you to move/pan the displayed data objects in the viewer.

Zoom In: Allows you to zoom in on the displayed data objects.

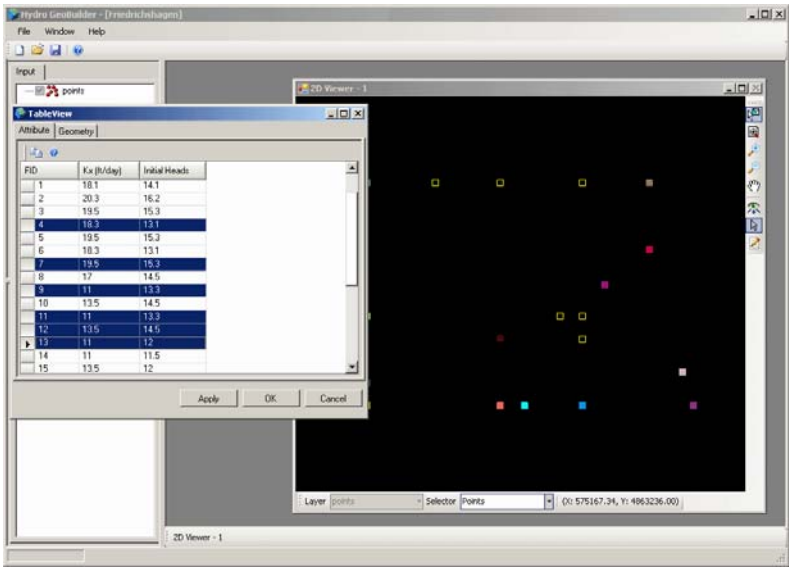
Zoom Out: Allows you to zoom out of the displayed data objects.

Zoom Into box:: Use the mouse cursor to draw a box around an area of interest, and automatically zoom into this area.

5.6 Linking 2D Viewers with Attribute Tables

The 2D Viewer can be linked with the spreadsheet table such that when a *polyline*, *polygon* or *point* feature is selected in the 2D viewer, its corresponding attribute data is automatically highlighted in the spreadsheet table. Likewise, when an attribute row is selected in the spreadsheet table, its corresponding feature is highlighted in the active 2D Viewer. In order to have this bidirectional linking between viewer and spreadsheet table, the 2D Viewer must be in **Pick** mode, and the spreadsheet table must be opened.

Tip! Data object spreadsheet tables can be viewed by right-clicking on the data object in the **Data Explorer**, and selecting **Spreadsheet...** from the pop-up menu. 2D Viewers can be set to pick mode by selecting the **Pick Mode** button from the viewer sidebar.



5.7 Exporting Viewers

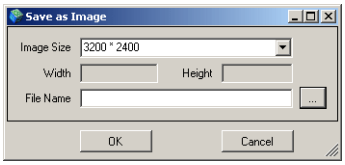
To export a **2D Viewer** to a graphics file, *e.g.*, *.BMP, .*TIF, *.GIF, *.JPEG, follow the steps below:

- Right-click anywhere within the viewer
- Select **Export Current View to Image** from the pop-up menu.
- A **Save As** window will display, allowing to you specify the location on your computer where the graphics file will be saved.

To export a **3D Viewer** to a graphics files, follow the steps below:

- Right-click anywhere within the viewer

- Select **Save as image** from the pop-up menu.
- The following dialog box will display on your screen:

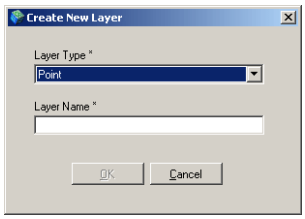


- Select the desired image size from the **Image Size** combo box. If you select **Custom**, then specify the desired image dimensions in the **Height** and **Width** combo boxes.
- Click the **[...]** button and specify a folder location on your computer to save the image file.
- Click the **[Ok]** button to save the image.

5.8 Creating a New Data Object

The 2D Viewer provides interactive drawing tools which allow you to create your own polygon, polyline and point data objects. This feature can be useful for digitizing boundary condition areas, property zones or your conceptual model boundary. To create a new polygon, polyline or points data object, follow the steps below:

- In the **Data Explorer**, right-click and select **Create New Data Object** from the pop-up menu. The following dialog will display:



- Select the data object type from the **Layer Type** combo box.
- Enter a name for the data object in the **Layer Name** field.
- Click the **[Ok]** button to create the new data object.

Once created, the new data object will appear in the **Data Explorer**. From here, you can define the geometry of the data object using the 2D Viewer editing tools. These tools are described in the following section.

5.9 Digitizing & Editing Geometry in 2D Viewers

The 2D Viewer allows you to create and modify the geometry of *points*, *polylines* and *polygon* data objects. The process of drawing in a 2D Viewer is described below:

- Open a **2D Viewer** by selecting **Window** from the Hydro GeoBuilder main menu, and then clicking **New 2D Window**.
- Next, display the data object that will be edited in the 2D Viewer. **Note**: You can have multiple data objects displayed in the viewer while you're editing/ drawing the data object. However, make sure that the data object being edited is the "active" one by selecting the data object name from the **Layer** combo box, located at the bottom of the 2D Viewer window.
- From the 2D Viewer sidebar, select the **Pick** button to set the 2D Viewer to pick mode. Pick mode allows you to click and select individual shape elements, *e.g.*, vertices, line segments, features, that comprise the active data object.
- From the 2D Viewer sidebar, select the **Edit** button to set the 2D Viewer to edit mode. Once this button is selected, a set of editing buttons will display in the 2D Viewer sidebar. The edit buttons that show in the sidebar will vary depending on which type of data object is being edited. For example, the Add Points button will not be shown when you are editing/creating a polygon or polyline data object. The edit buttons are described below.

Add Points: Digitize points in the 2D Viewer by moving the mouse cursor to the desired location and clicking the left mouse button. This button only shows when creating/edit a Points data object.

Add Polyline: Digitize a polyline in the 2D Viewer. Click the left-mouse to start the line, and then left-click to insert a vertex along the line path. Double-click to end the polyline. This button only shows when creating/editing a Polyline data object.

Add Polygon: Digitize a polygon in the 2D Viewer. Click the left-mouse button to start the polygon. Each successive left mouse-click will insert a vertex. Double-click to close the polygon..

Move Points: Move a point by selecting and dragging the points to a new location in the 2D Viewer.

Rescale: Select a shape element and stretch or shrink the geometry by selecting and dragging a side or corner of the blue box.

Rotate: Select a shape element and rotate the geometry clockwise or anti-clockwise by selecting and holding the blue box, while moving the mouse.

Delete Shape: Delete the selected shape.

Undo All: Undo all edits. This button will revert the data object back to its original geometry.

- If you are editing an existing polyline or polygon data object, the **Selector** combo box located at the bottom of the 2D Viewer allows you to select and modify the points (vertices) that comprise the features in the data object. For example, if you are editing a polyline data object, you can select **Points** from the **Selector** combo box, and each vertex that comprises each polyline will become active, allowing you to add, move or delete the vertices. When the **Selector** combo box is set to **Points**, the following icons are added to the 2D Viewer sidebar.

Add Vertex: Add a vertex to a polyline or polygon feature by placing your mouse in the desired location on the line or polygon boundary, and clicking the left mouse button.

Move Vertex: Select and hold the left mouse button and move the vertex to a new location the line or polygon boundary.

Delete Vertex: When selected, select a vertex to remove from a line and or polygon boundary.

- Once you have created/modified the data object geometry using the edit tools described above, click the **End Edit** button to save the changes.

Finally, click the **View** button to return to the normal 2D Viewing mode.

6

Creating Surfaces

In Hydro GeoBuilder, a surface refer to an attribute, *e.g.*, elevation, conductivity, heads, represented as a set of continuous data over an area. Surfaces can either be imported directly (see “Importing Surfaces” on page 33) from various file types using the import utility, or created by interpolating one or more points data objects. This chapter describes the process of creating surfaces from points data objects.

Surfaces are required in Hydro GeoBuilder for defining the vertical boundaries of structural zones, *i.e.*, horizons, in a conceptual model. However, surfaces can also be used to assign spatially-variable attributes to property zones and boundary conditions, or for simply visualizing spatial variation using the 2D or 3D Viewers.

The create surface process allows you to generate surfaces using any numeric attribute in a point data object. The point data object can be one that has been imported using the import utility, or one that has been generated from other data objects, *e.g.*, cross sections and wells. For more information on creating points data objects from well data and cross sections, please see “Converting Well Tops to Points Data Object” on page 74 or “Converting Model Layers to Points Data Object” on page 74, respectively.

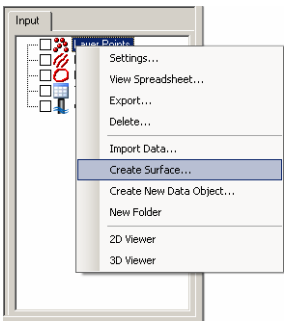
This chapter presents information on the following topics:

- Creating a New Surface
- General Settings
- Interpolation Settings
- Deleting a Surface

6.1 Creating a New Surface

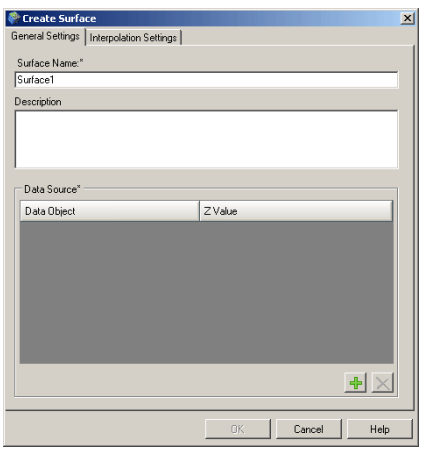
To create a new surface, follow the steps below:

- From the **Data Explorer**, right-click anywhere and select **Create Surface...** from the pop-up menu.



- The **Create Surface** dialog will display. Specify the general settings, described in the following section

General Settings

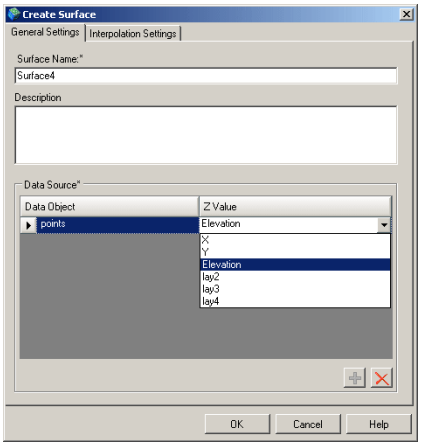


Enter a unique name for the surface in the **Surface Name** field, and a description of the surface (optional) in the **Description** text box.

The **Data Source** frame allows you to select the point data object(s) from which the surface will be created.

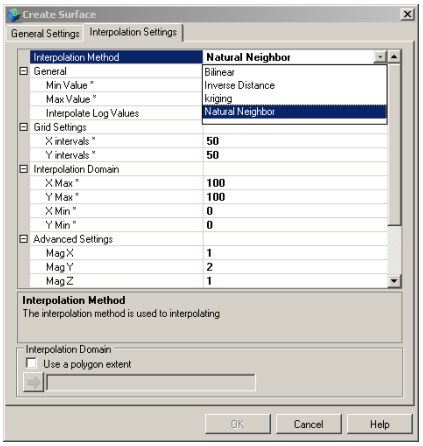
Adding a Data Source

- From the **Data Explorer**, select the desired **Point** data object from which the surface will be interpolated.
- Click the **Add** button, to add the point data object to the **Data Source** frame.
- Hydro GeoBuilder allows you to create a single surface from multiple point data objects. Repeat the steps above to add additional points data objects to the **Data Source** frame.
- For each data source, select the **Z Value** from the combo box (shown below). The Z value can be any numeric attribute stored in the points data source, *e.g.*, Elevation, Conductivity, etc.



Next, click the **Interpolation Settings** tab to define the interpolation settings for the surface. These settings are described in the following section.

Interpolation Settings



Select the interpolation method to use for generating the surface. Choose from the following interpolation methods:

- **Inverse Distance**
- **Kriging**
- **Natural Neighbor**

Below are brief descriptions of each interpolation method, taken from the GSLIB Geostatistical Software Library and User's Guide (Deutsch and Journel, 1998). For a description of each interpolation parameter setting, click on the setting, and a brief description will be displayed beneath the interpolation settings grid.

Once the settings have been defined, click the **[OK]** button to generate the surface.

Inverse Distance

The Inverse Distance Squared method is very fast and efficient, weighted average interpolation method. The weighting factor applied to the data depends on the distance of the point from the grid cell, and is inversely proportional to the distance squared. Consequently, the greater the distance the data point is from the grid node, the smaller the influence it has on the calculated value.

The Inverse Distance Squared method for interpolation may generate patterns similar to the "bull's-eye" surrounding points of observations. Selecting a larger number of nearest neighboring data points may smooth this effect, but if the bull's eye pattern is undesirable, then other methods for interpolation, like Natural Neighbor and Kriging, are recommended.

Kriging

Kriging is a geostatistical method that produces visually appealing maps from irregularly spaced data. Anistropy and underlying trends suggested in raw data can be incorporated in an efficient manner through Kriging. The program used, called kt3d, is available in the public domain from the Geostatistical Software Library (GSLIB), distributed by Stanford University, and is well documented by Deutsch and Journel (1998). The project kt3d performs simple Kriging, ordinary Kriging, or Kriging with a polynomial trend, and uses the standard parameter file used by GSLIP. If the semi-variogram components have already been modeled by the user, they can be incorporated into the program by choosing the appropriate set of parameters in the parameter file. The semi-variograms available include Spherical, Exponential, Gaussian, Power, and the Hoe effect models. If the variogram information is not available, the default linear variogram with no nugget effect should be used. This option is a special case of the Power model with the exponent equal to 1.

Natural Neighbors

The Natural Neighbor method (Watson, 1994) is based on the Thiessen polygon method used for interpolating rainfall data. The grid node for interpolation is considered a new point, or target, to the existing data set. With the addition of this point, the Thiessen polygons based on the existing points are modified to include the new point. The polygons reduce in area to include the new points, and the area that is taken out from the existing polygons is called the “borrowed area”. The interpolation algorithm calculates the interpolated value as the weighted average of the neighboring observations where the weights are proportional to the borrowed areas. The Natural Neighbor method is valid only with the convex hull of the Thiessen polygon formed by the data points, and values outside the hull extrapolation should be used with caution.

The Natural Neighbor interpolation scheme may be visualized as a taut rubber sheet stretched to satisfy all the data points. The interpolated value at any location is a linear combination of all Natural Neighbors of that location, and the resulting surface is continuous with a slope that is also continuous. Combining the gradients or slopes with the linear interpolation provides results that are more smooth, and may anticipate the peak and valleys between data. Singularities and other undesirable effects may be lessened by incorporating the gradient factor.

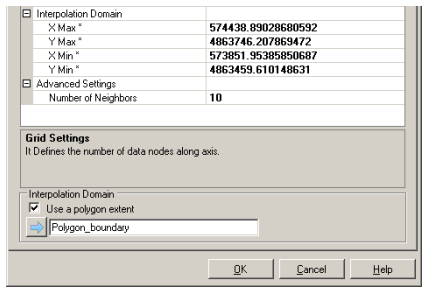
The gradient influence on the results can be manipulated by two tautness parameters that you can enter. These parameters allow the interpolated surface to vary from purely linear interpolation to one which is well rounded and has a gradient factor. In all cases the slope discontinuities are removed and the resulting surface has slope continuity everywhere.

Defining an Interpolation Domain using a Polygon

When creating a surface, the interpolation domain is automatically calculated from the X and Y extents of the specified point(s) data set. There may be times when you do not want to use the entire points data set to generate a surface. In this case, you can manually define the interpolation domain (Xmax, Ymax, Xmin, Ymin) by specifying new values in the interpolation settings grid, or you can use an imported or digitized polygon data object.

To define the interpolation domain using a polygon data object, follow the steps below:

- In the **Create Surface** dialog box, select the **Use a polygon extent** check box.
- Select a polygon data object from the **Data Explorer**, and then click the button.



When the polygon data object is selected, the interpolation domain values in the settings grid will update with the X-Y extents of the selected polygon data object.

6.2 Surface Settings

Once a surface data object is generated, you can view and modify various settings by right-clicking on the data object, and selecting **Settings...** from the pop-up menu.

For more information on surface settings, please see “Surfaces” on page 88.

6.3 Deleting a Surface

To delete a surface data object, right-click on the data object in the **Data Explorer**, and select **Delete...** from the pop-up menu. Please take caution when deleting data objects, as this operation cannot be undone.

Note: Surface data objects that are used to define conceptual model horizons cannot be deleted from the project. A warning message will appear to indicate that the surface is used for the horizon.

7

Creating a Conceptual Model

A **Conceptual Model** refers to a basic, high-level representation of the hydrogeological system being modeled. Once you have imported sufficient raw data into your project, you can begin to construct one or more conceptual models using imported or digitized data objects as building blocks.

Note: The minimum data requirements for building a conceptual model are **two surfaces**, and **one polygon** data object.

In general, a groundwater flow conceptual model is comprised of the following sub-models:

- **Structural Model:** Consists of a polygon that represents the horizontal boundary of the model, and a series of horizons that represent the vertical boundaries of subsurface structural zones. Together, these boundaries represent the simulation model domain. For more information on structural modeling, please see Chapter 8: Defining Horizons.
- **Property Model:** Consists of property zones generated by combining or subdividing structural zones. Each property zone is assigned appropriate property attributes, *e.g.*, conductivity, storage, and initial heads. For more information on property modeling, please see Chapter 9: Property Modeling.
- **Boundary Condition Model:** Consists of defining flow conditions at the boundaries of the conceptual model, *e.g.*, recharge, river, lake, specified head etc. For more information on defining boundary conditions, please see Chapter 11: Boundary Modeling.

When a new conceptual model is created, a new conceptual model tree is added to the **Conceptual Model Explorer**, in the main Hydro GeoBuilder window. The conceptual model tree consists of a fixed folder structure that is designed to guide you through the workflow of building your conceptual model.

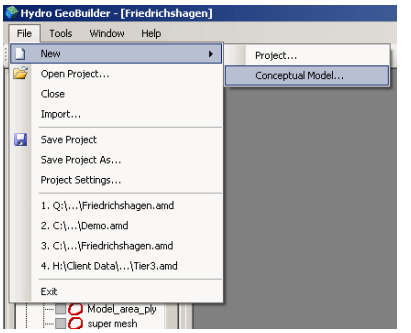
This chapter presents information on the following topics:

- Creating a New Conceptual Model
- Conceptual Model Settings
- Conceptual Model Tree
- Deleting a Conceptual Model

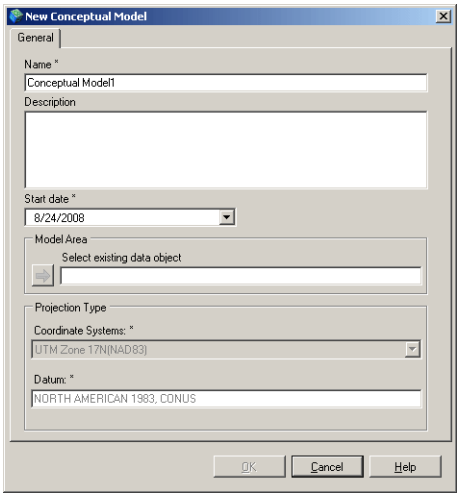
7.1 Creating a New Conceptual Model

To create a new conceptual model, follow the steps below:

- From the Hydro GeoBuilder main menu, select **File** > **New** > **Conceptual Model...**



- The **Create New Conceptual Model** dialog box will launch (shown below) where you can define the settings for the conceptual model.



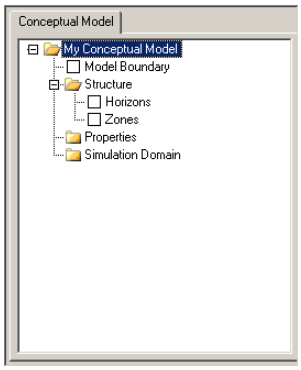
- Enter a unique name for the conceptual model in the **Name** field.
- Enter a description of the conceptual model in the **Description** field (optional).
- Specify a start date from the **Start Date** combo box.
- From the **Data Explorer**, select the polygon data object that represents the conceptual model horizontal boundary, and then click the button.

Note: The model area cannot be defined using a complex polygon, or one that contains multiple polygons. A complex polygon is a polygon that intersects with itself.

- The **Coordinate System** is automatically set to the project's coordinate system and cannot be modified.
- Click the **[Ok]** button to save the settings and to create the conceptual model.

Conceptual Model Tree

Once a conceptual model is created, a new conceptual model tree is added to the **Conceptual Model Explorer**. The conceptual model tree sets up the workflow for structural and property modeling, assigning boundary conditions, numerical grid creation, and numerical model translation. A typical conceptual model tree is shown below:



The **Model Boundary** node allows you to show/hide the conceptual model boundary in a 2D or 3D Viewer.

The **Structure** folder allows you to define the horizons and structural zones of the conceptual model. For more information on structural modeling, please see "Defining Horizons" on page 121.

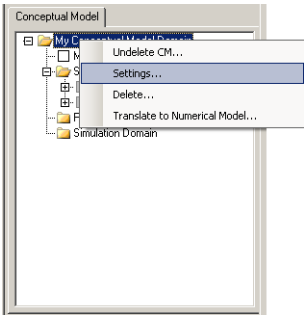
The **Properties** node allows you to define property zones for the conceptual model. For more information on property modeling, please see "Property Modeling" on page 129.

The **Simulation Domain** node allows you to define the simulation model domain, assign boundary conditions and generate numerical model grids. For more information on these topics, please see Chapter 10: Simulation Model Domain, Chapter 11: Boundary Modeling, Chapter 12: Model Domain Discretization, respectively.

7.2 Conceptual Model Settings

To view and modify the conceptual model settings, follow the steps below:

- From the **Conceptual Model** tree, right-click on the root of the conceptual model tree, and select **Settings...** from the pop-up menu.



The **Conceptual Model Settings** dialog box will open. Here you can modify the conceptual model **Name**, **Description**, and **Start** date. You **cannot** select a new polygon for the model boundary. If you wish to use a different polygon for the model boundary you must create a new conceptual model.

7.3 Deleting a Conceptual Model

To delete a conceptual model, right-click on the root of the conceptual model tree, and select **Delete...** from the pop-up menu. You will be prompted with a confirmation message before the conceptual model is deleted. Click the **[Yes]** button to delete the conceptual model.

Note: Please be aware that there is no undo function to recover a deleted conceptual model. Please exercise caution when deleting conceptual models.

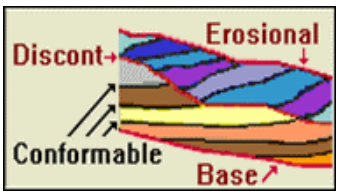
8

Defining Horizons

Horizons are stratigraphic layers (2D surfaces with topography) that define the upper and lower boundaries of the structural zones in a conceptual model. In Hydro GeoBuilder, horizons are created by clipping or extending interpolated surface data objects to the boundary of the conceptual model. For more information on creating surface data objects, please refer to Chapter 6: Creating Surfaces.

When horizons are created, Hydro GeoBuilder will automatically generate the **Structural Zones** (geologic formations) between the horizons, which can be used later to define property zones.

Horizon Types



Each horizon can be assigned a particular type, which defines the relationship to other horizons in the conceptual model. This prevents intersecting layers and establishes layers that satisfy both FEFLOW and MODFLOW requirements. Each horizon type is described below:

Erosional horizons can be used as the highest or as an intermediate horizon, but not as the bottom of the conceptual model. This type of horizon will truncate all horizons below it, including the base horizon.

Base horizons can be used as the lowest horizon in the conceptual model. Any conformable horizon types will lap onto it, while all erosional or discontinuity horizons will truncate it.

Discontinuity horizons represent an erosional surface in the middle of a stack of horizons. It can never be the highest or lowest horizon. Horizons above it up to the next

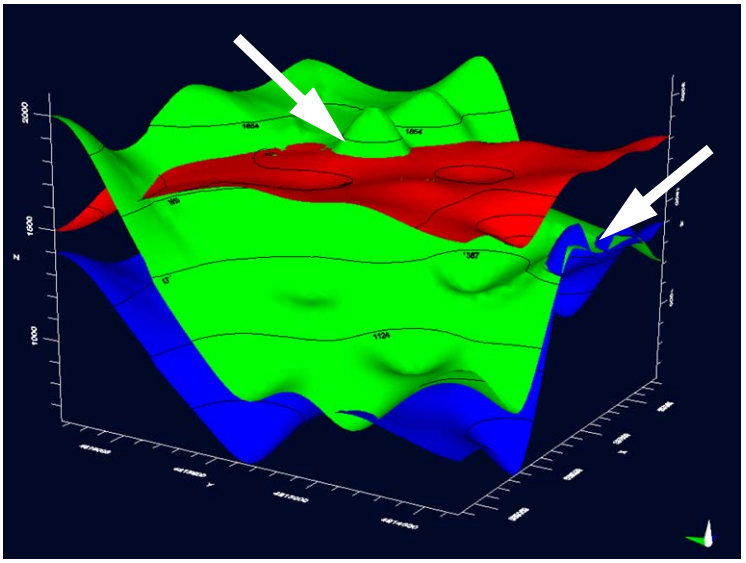
discontinuity or erosional horizon will lap onto it, while all horizons below it will be truncated by it. These horizons can be thought of as the top or base of a sequence.

Conformable (default) horizons will be truncated by erosional, base and discontinuous horizons. Lower conformable horizons will be truncated by upper conformable horizons. If a conformable horizon is above an erosional horizon, the conformable horizon will “conform” to the erosional horizon (it will be pushed up by the erosional horizon).

The horizon rules described above are applied after all the horizons are calculated. If one of the horizons will be truncated by an erosional, base, or discontinuity horizon, it is a good idea to extend the input data beyond these unconformable horizons in order to truncate them properly.

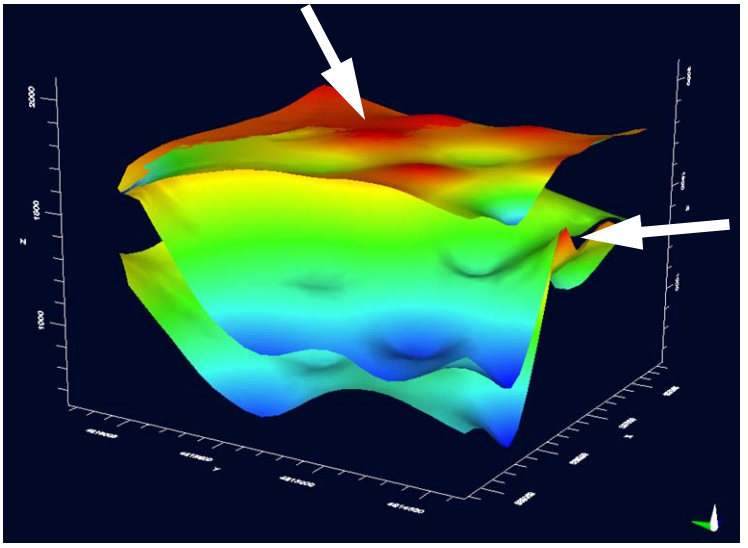
Horizon Types Example

The image below shows three surfaces in a 3D Viewer. The surfaces are colored Red, Green, Blue, from top to bottom, respectively.

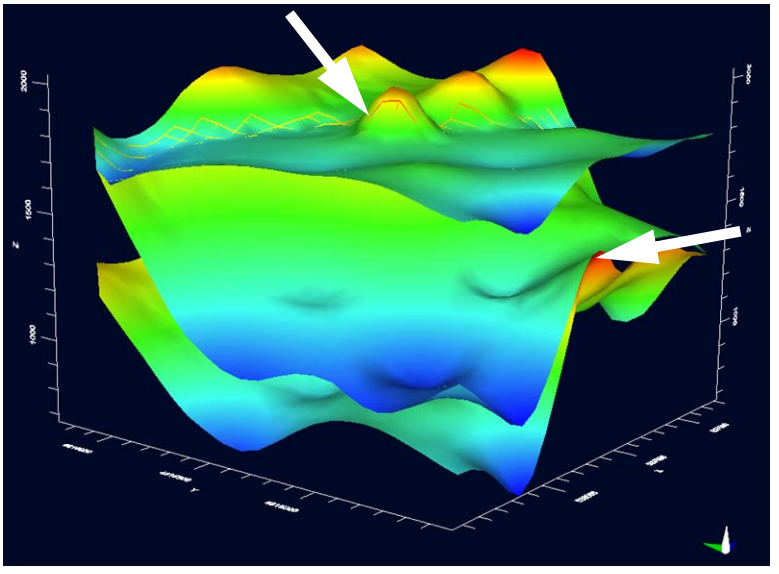


You will see that there are spots where the green surface intersects with the red surfaces, and likewise, where the blue surface intersects with the green surface. For numerical models (FEFLOW and MODFLOW), this geometry is not permitted which is why surfaces are converted to horizons. Assigning a horizon type will eliminate the intersections.

When horizons are generated from these surfaces, and each surface is set to **Conformable**, each horizon is truncated that each conforms to the horizon above it, as shown in the following image.



If the middle horizon is set to **Erosional**, with the top and bottom set to **Conformable**, the topmost surface (red) is pushed up, as shown in the following image.

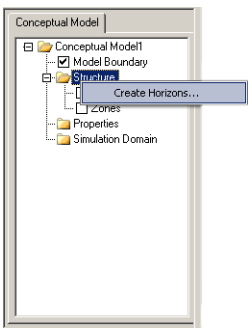


8.1 Creating Horizons

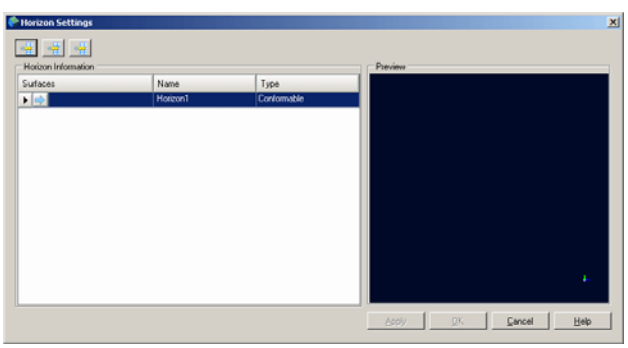
Note: In order to define horizons, you must have already created a new conceptual model. For information on creating a conceptual model, please refer to see “Creating a Conceptual Model” on page 117.

To create a new horizon, follow the steps below:

- From the **Conceptual Model Tree**, expand the **Structure** node, right-click on **Structure** and select **Create Horizons...**



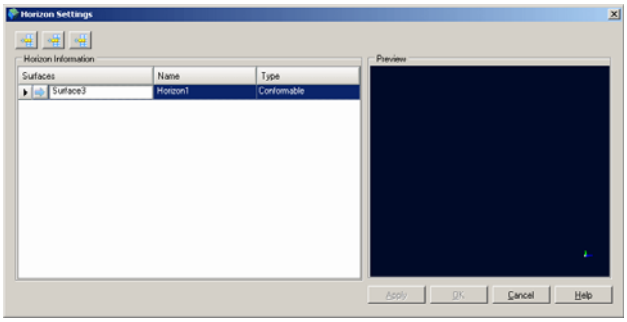
- From the **Horizons Settings** dialog (shown below), click the **Add Horizon** button to add a new horizon row to the **Horizon Information** table.



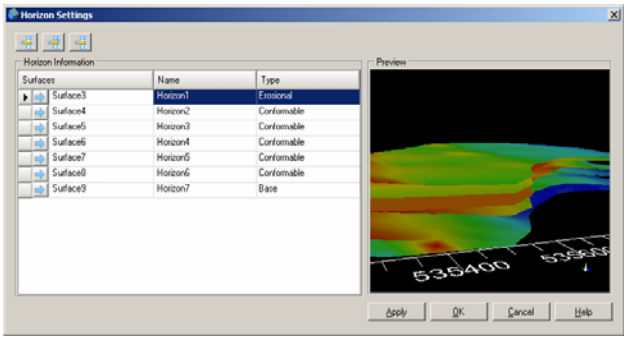
- From the **Data Explorer**, select the surface data object that will be used to generate the horizon, and then click the **Blue Arrow** button to insert it into the **Horizon Information** table. If the selected surface is greater than the conceptual model area, it will be clipped by the conceptual model boundary

polygon. If the surface is less than the conceptual model area, it will be extended to the conceptual model boundary.

Note: Surfaces should be added according to elevation, starting with the top surface (ground surface) and ending with the bottom surface.

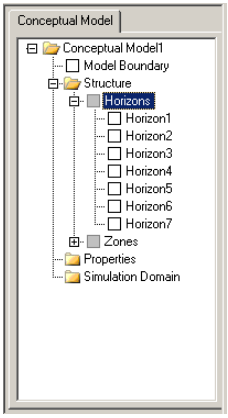


- In the **Name** column, type in a unique name for the horizon.
- In the **Type** column, select the appropriate horizon type from the combo box. For information on each horizon type, please refer to see “Horizon Types” on page 121.
- Repeat the steps above to add additional horizons. Remember you must have at least two horizons before Hydro GeoBuilder can create the structural zones.
- You can preview the horizons in the adjacent 3D Viewer, by clicking the **[Apply]** button.



- Finally, click the **[Ok]** button to create the horizons.

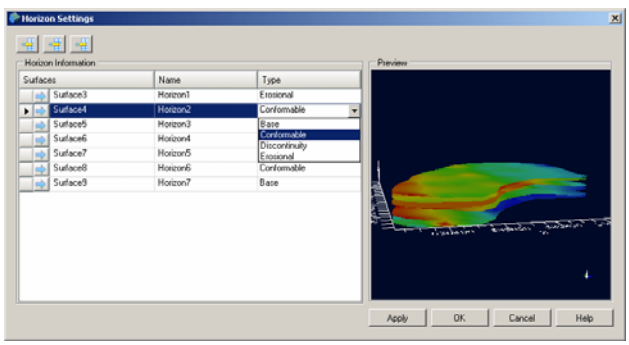
Once created, the horizons will be added to the **Conceptual Model Tree** under the **Horizons** node (shown below).



8.2 Editing and Deleting Horizons

Once horizons are created, you can make modifications to the horizon settings, add and insert new horizons, or delete horizons. To do so, follow the steps below:

- From the **Conceptual Model Tree**, expand the **Structure** node, right-click on the **Horizons** node, and select **Horizon Settings....**



In the **Horizon Settings** dialog, you can edit the **Source**, **Name**, and **Type** of the existing horizons.

You can **Add** or **Insert** a new horizon to the **Horizon Information** table by clicking on the appropriate button at the top of the dialog.

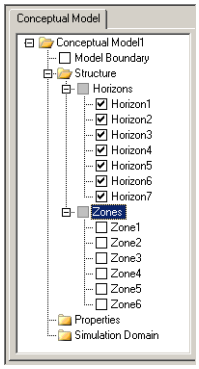
To delete a horizon, click on the grey box located to the left of the blue arrow, for the horizon that you want to delete. Once the row is highlighted blue, as shown in the image above, click the **Delete** button.

Note: The delete operation cannot be undone. Any deleted horizons must be recreated.

Once the changes are made to the **Horizon Settings**, you can preview the modifications in the adjacent 3D Viewer by clicking on the **[Apply]** button. Otherwise, click the **[Ok]** button to save the settings.

8.3 Viewing Structural Zones

During the horizon creation process, Hydro GeoBuilder automatically generates the structural zones between the defined horizons within the horizontal extent of the conceptual model boundary. To view the generated structural zones, in the **Conceptual Model Tree**, expand the **Structure Node** and then expand the **Zones** node**.**



Zones are given a default name, *e.g.*, Zone1, Zone2, Zone3 etc., which cannot be modified.

You can view the zones in an active 3D Viewer window by checking the empty check box beside the zone name. For more information on data viewers, please refer to “Data Viewers” on page 95.

9

Property Modeling

A groundwater flow model requires many different types of data to simulate the hydrogeological processes influencing the flow of groundwater. In Hydro GeoBuilder, the hydrogeological characteristics of the model are classified into the following parameter groups:

- Conductivity (K_x, K_y, K_z)
- Storage (S_s, S_y, P_{eff}, P_{tot})
- Initial Heads

By default, Hydro GeoBuilder automatically assigns the entire model domain the default property parameter values, specified in the **Project Settings** (see “Modifying Project Settings” on page 18) **.** However, in most situations, the flow properties will not be uniform throughout the entire model domain, and it will be necessary to assign different property values to different areas of the conceptual model. This can be accomplished by creating **Property Zones**. In Hydro GeoBuilder, a property zone is a specified 3D volume, generated from structural zones, with user-defined hydrogeologic attributes.

Property zone geometry can be defined using one or more existing structural zones. As such, property zones can only be generated after horizons have been defined in the conceptual model. For more information on defining horizons, please see Chapter 8: Defining Horizons.

Hydro GeoBuilder supports various methods for assigning values to hydrogeologic parameters. The method used for defining attributes can be defined on the parameter level, allowing you to use different methods for different parameters. The supported methods include:

- Use Constant Value
- Use Surface Data Object
- Use 3D Gridded Data Object
- Use Shapefile

The following sections provide information on the following topics:

- Defining a New Property Zone

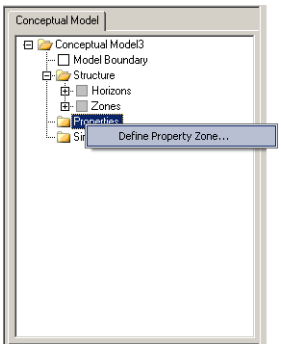
- Assigning Property Parameters
- Editing Property Zones
- Deleting a Property Zone

9.1 Defining a New Property Zone

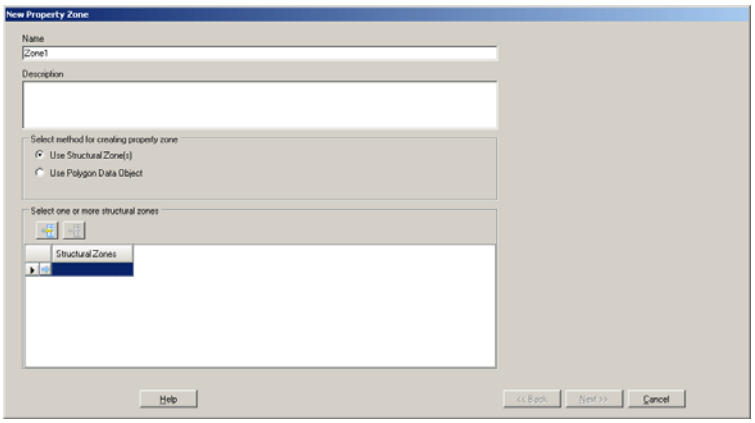
Before you can create a property zone, you must have already defined horizons for the conceptual model. For more information on defining horizons, please see Chapter 8: Defining Horizons.

To define a new property zone, follow the steps below:

- Right-click on the **Properties** node in the **Conceptual Model Tree**, and select **Define Property Zone...**.



- The **New Property Zone** dialog will display (shown below). Enter a **Name** and **Description** (optional) for the property zone in the **Name** and **Description** fields, respectively.

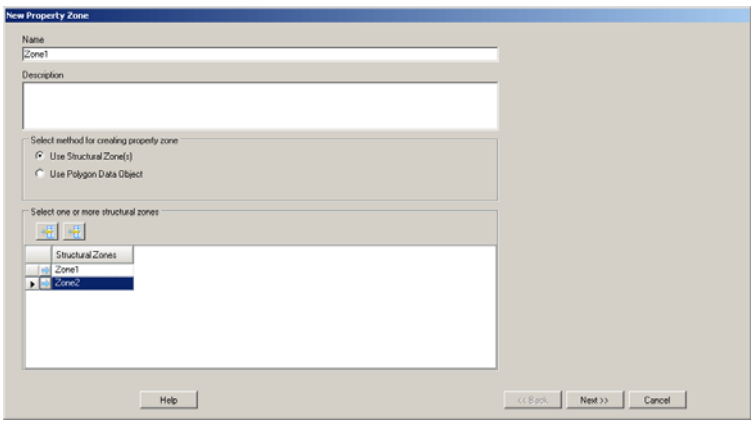


Select the method by which the property zone geometry will be defined. There are two options: **Use Structural Zone(s)** and **Use Polygon Data Object**.

Selecting a Method for Defining Property Zones

Using Structural Zone(s)

This method allows you to create a property zone from existing structural zones in your conceptual model, *i.e.*, zones generated from horizons.

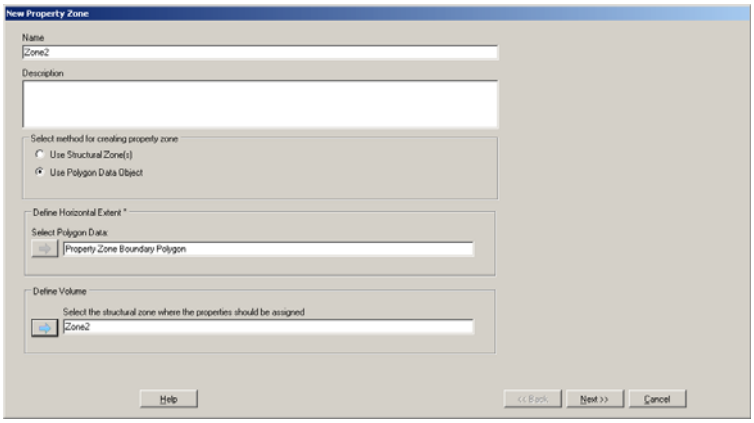


Select a zone from the conceptual model tree (under the **Zones** node), and then click the button to insert the zone in the **Structural Zones** field. Click the **Add** button, to add and combine structural zones. To delete a structural zone from the grid, select the structural zone, and click the **Delete** button.

Once the desired structural zones are selected, click the **[Next]** button to proceed to the next step, which is described in “Assigning Property Parameters” on page 132.

Using Polygon Data Object

This method allows you to define a property zone using both a structural zone and a polygon data object. The polygon data object is used to define the horizontal extent of the property zone and therefore must be fully contained within the conceptual model boundary. The structural zone is used to define the volume, *i.e.*, the vertical extent of the property zone.



Select a polygon data object from the **Data Explorer**, and click the button to insert the data object in the **Select Polygon Data** field.

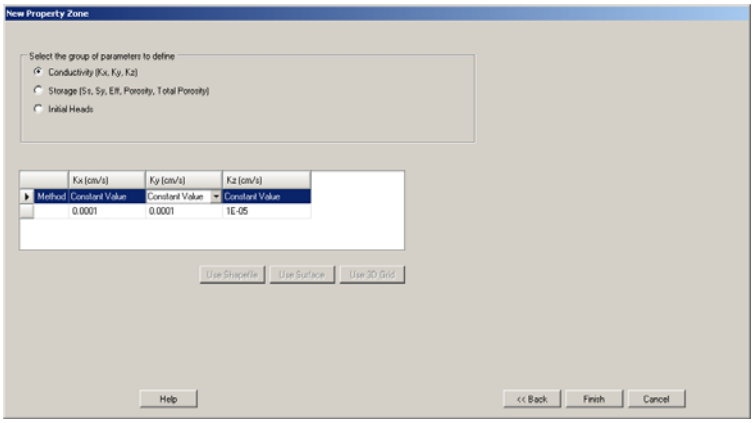
> **Note:** The selected polygon cannot contain multiple parts, overlapping shapes or holes. These features are currently not supported for property zone creation. If your polygon does not meet this criteria, it can be edited using the 2D Viewer editing tools. For more information on this topic, please see “Digitizing & Editing Geometry in 2D Viewers” on page 107.

Next, select a structural zone from the **Conceptual Model** tree, and click the button to insert the data object in the **Define Volume** frame.

Click the **[Next]** button to proceed to the next step.

Assigning Property Parameters

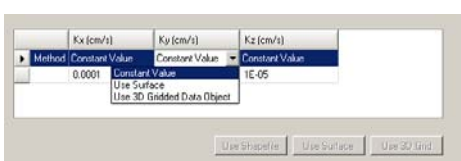
Once the geometry has been defined, you can assign parameter values to the property zone.



First, select the group of parameters that will be defined, *e.g.*, conductivity, storage or initial heads. The data input grid below will display the appropriate parameters based on which parameter group is selected. For example, if conductivity is selected, the data input grid will show the parameters K_x, K_y, and K_z. The data input grid will already be populated with the default values specified in the **Project Settings** (**File > Project Settings...**).

Note: If the property zone is defined using a polygon data object, and the data object consists of multiple polygons, you can assign unique parameters to each individual polygon, or you can assign parameters defined for one polygon to all polygons by clicking the **[Apply to All..]** button.

Hydro GeoBuilder provides various methods for assigning parameter attributes. The available methods include: **Constant Value**, **Use Surface**, **Use 3D Gridded Data** and **Use Shapefile** (available only when property zone is defined using polygon data object). The type of method used can be specified per parameter. For each parameter in the data input grid there is a combo box in the **Method** row (shown below).



For each parameter in the data input grid, specify the method for defining attribute values by selecting the desired option from the **Method** combo boxes.

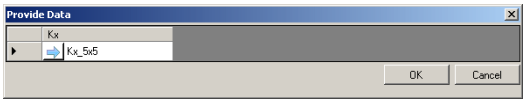
Each method is described in the following sections.

Constant Value

The **Constant Value** method is selected by default for each parameter in the data input grid and allows you to specify a spatially constant value for the parameter. If you do not wish to use the default value, enter a new value.

Use Surface

The **Use Surface** method allows you use an existing surface data object to define spatially-variable attribute values. When this method is selected, the **Use Surface** button will become active. Click the **Use Surface** button to launch the **Provide Data** dialog box (shown below).

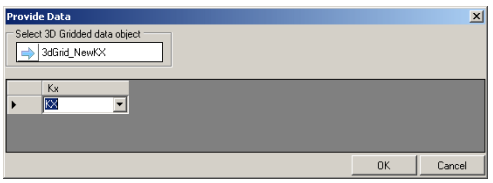


From the **Data Explorer**, select the desired surface data object and then click the button to insert it into the parameter field. Click the **[OK]** button to close the dialog box.

Note: The selected surface data object must cover the entire area of the property zone, or else the data object cannot be used.

Use 3D Gridded Data

The **Use 3D Gridded Data** method allows you to use an existing 3D gridded data object to define spatially-variable attribute values. When this method is selected, the **Use 3D Grid** button will become active. Click the **Use 3D Grid** button to launch the **Provide Data** dialog box (shown below).

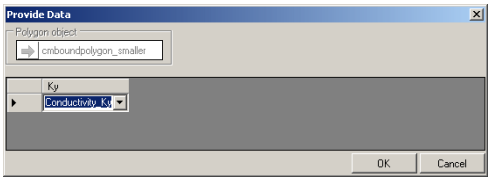


From the **Data Explorer**, select the desired 3D Gridded data object and then click the button to insert it into the **Select 3D Gridded Data Object** field. Once selected, the data objects attributes are listed in the combo box below. Select the desired attribute value from the combo box, and then click the **[Ok]** button to close the dialog box.

Note: The specified 3D Gridded data object must horizontally and vertically overlap the defined property zone geometry, or else the data object cannot be used.

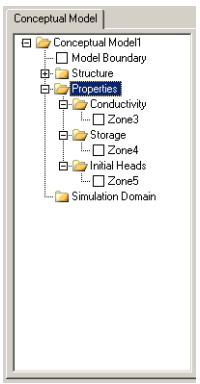
Use Shapefile

The **Use Shapefile** method is only available when you define the property zone geometry using a polygon data object. This method allows you to assign an attribute value using an attribute from the specified polygon data object. When this method is selected, the **Use Shapefile** button will become active. Click the **Use Shapefile** button to launch the **Provide Data** dialog box (shown below).



The combo box contains all the attributes of the specified polygon used to define the horizontal geometry of the property zone. Select the desired attribute from the combo box, and then click the **[Ok]** button to close the dialog box.

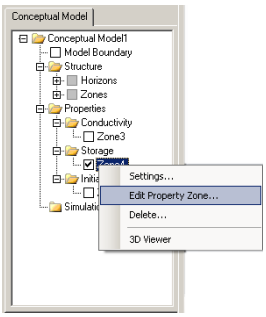
Once the property zone attributes have been defined, click the **[Finish]** button to create the property zone. Once created, the property zone is added to the **Conceptual Model** tree under the **Properties** node and under the appropriate parameter category node.



9.2 Editing Property Zones

Once a property zone has been created, you can go back and modify the property zone name and attribute values. To do so, follow the steps below:

- From the **Conceptual Model** tree, expand the **Properties** node, and the desired property category node. Right-click on the property zone and select **Edit Property Zone...** from the pop-up menu.



- The **Edit Property Zones Settings** dialog box will display on your screen. Here you can change the **Name** of the property zone. You cannot change the geometry of the property zone.
- Click the **[Next]** button to advance to the property parameters dialog. Here you can modify the parameter values for the property zone. These options are described in “Assigning Property Parameters” on page 132.
- Click **[Finish]** button to apply the modifications to the property zone.

9.3 Property Zone Settings

To access the settings for a property zone, right-click on the desired property zone, and select **Settings...** from the pop-up menu.

The settings for property zones only include general read-only meta data information, *e.g.*, name, description, data source and coordinate system, as well property zone color settings (for displaying in 3D Viewers). These general settings are common to other data objects, and are described in the section “General” on page 63.

9.4 Deleting a Property Zone

To delete a property zone, follow the steps below:

- From the **Conceptual Model** tree, expand the **Properties** node, and the desired property category node. Right-click on the property zone and select **Delete...** from the pop-up menu.
- A confirmation message will display on your screen. Click the **[Yes]** button to confirm and delete the property zone. **Note: This operation cannot be undone.**

10

Simulation Model Domain

The **Simulation Model Domain** is the total area and volume of the conceptual model from which the numerical model will be generated. Its horizontal boundaries are defined using the conceptual model domain boundary, and its vertical boundaries are defined using the upper-most and lower-most horizons. The simulation model domain can only be generated after you have created the appropriate horizons and structural zones for the conceptual model (see Chapter 8: Defining Horizons).

Note: Once the simulation model domain is generated, any changes made to the model horizons and structural zones will **not** be reflected in the simulation model domain (numerical model). If changes to the structural model is required after the simulation model domain has been created, you must create a new conceptual model, and regenerate the simulation model domain.

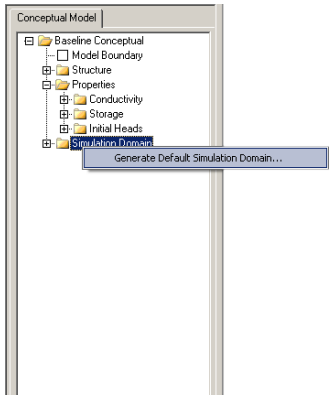
Once the simulation model domain is generated, you can assign appropriate boundary conditions to the top, bottom, intermediate layers or sides of the simulation domain. By default, the boundaries of the simulation domain are assigned a no-flow boundary condition. For more information on assigning boundary conditions to the simulation model domain, please see Chapter 11: Boundary Modeling.

This chapter describes the process of creating the simulation model domain.

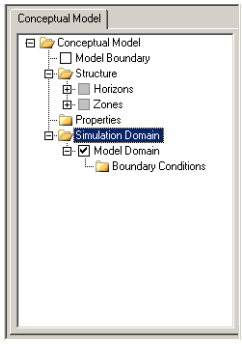
10.1 Creating the Simulation Model Domain

To create a new simulation domain, follow the steps below:

- From the **Conceptual Model Tree**, right-click on the **Simulation Domain** folder, and select **Generate Default Simulation Domain...** from the pop-up menu.



Once the simulation model domain is generated, a subnode called **Model Domain** is created under the **Simulation Domain** folder in the **Conceptual Model** Tree (shown below).



The **Model Domain** node allows you to view the simulation model domain as a 3D object in a 3D Viewer by selecting the adjacent check box. Right-clicking on the Model Domain node allows you to view the **Settings** of the simulation model domain (see Chapter 4: Data Settings and Properties) and create **Numerical Model Grids** (see Chapter 12: Model Domain Discretization)

Below the **Model Domain** node is another subfolder called **Boundary Conditions**. This folder allows you to assign various boundary conditions to the simulation model domain (see Chapter 11: Boundary Modeling).

11

Boundary Modeling

Every conceptual model requires an appropriate set of boundary conditions to represent the system's relationship with the surrounding systems. In the case of groundwater flow model, boundary conditions will describe the exchange of flow between the model and the external system. Hydro GeoBuilder supports input and translation for various types of MODFLOW and FEFLOW boundary condition packages, including;

- Pumping Well
- Specified Head
- River
- General Head
- Drain
- Recharge
- Evapotranspiration
- Lake
- Specified Flux

Note: The MODFLOW Stream (STR) package is currently not supported in Hydro GeoBuilder

Note: Currently, only the **Pumping Well** and **Recharge** boundary conditions are supported for finite element model (FEFLOW) translation.

Boundary conditions can be applied to the top, bottom, intermediate layers or sides of the simulation model domain, using imported or digitized **Polygon** or **Polyline** data objects, or by **manually selecting** the sides of the simulation domain using an interactive 3D Viewer.

Hydro GeoBuilder provides various methods for assigning parameter values to boundary conditions. Each parameter in the boundary condition can be set to **Constant** or **Transient,** and values can be assigned using attributes from various imported data objects. The available methods for assigning attributes include:

- Using a Constant Value
- From Surface data object
- From Shapefile (Polygon or Polyline)
- From Time Schedule data objects

- From Spatial Attribute (3D Gridded Data) data object.

This chapter presents information on the following topics:

- Boundary Conditions Overview
- Defining a New Boundary Condition
- Defining a Pumping Well Boundary Condition
- Defining Other Boundary Conditions
  - Specifying Boundary Condition Type and Location
  - Specifying Boundary Condition Geometry (Horizontal)
  - Defining Polyline Zones
  - Defining Boundary Condition Geometry (Sides)
  - Defining Boundary Condition Parameters
- Modifying Boundary Conditions
- Deleting Boundary Conditions

11.1 Boundary Conditions Overview

The following sections present an overview of the boundary condition packages supported in Hydro GeoBuilder. Each section includes a brief description of the boundary condition, including the input data required by MODFLOW and the supported data objects for defining the boundary condition geometry. For information on how to create a boundary condition, please skip to “Defining a New Boundary Condition” on page 155.

11.1.1 Pumping Well

The pumping well boundary condition is used to simulate wells (or other features) that withdraw water from or add water to the model at a constant rate during a stress period, where the rate is independent of both the cell area and head in the cell.

For finite difference translations, Hydro GeoBuilder uses the Well (WEL) package, provided with MODFLOW. The MODFLOW input data for Well cells is stored in the *projectname*.WEL file. You can define the location for horizontal or deviated wells, which include the well path and the screen location. When you translate your conceptual model to MODFLOW format, the horizontal well screen location is converted to set of pumping well cells side-by-side. Another option is to define a specified flux or drain boundary condition in 3D-Builder. These are the workarounds, since there is no MODFLOW package for horizontal wells.

For finite element model translations, Hydro GeoBuilder translates the pumping well boundary conditions as a Type 4 (Well) boundary condition. Please note that the defined screen interval must extend beyond half of the element height for it to be assigned the

boundary condition.

Currently, deviated/horizontal well translation is not supported for finite element models. If you intend to translate to FEFLOW, please make sure all pumping well boundary conditions are defined using vertical wells.

Required Data

In Hydro GeoBuilder, pumping well boundary conditions are defined using the well data contained in a wells data object. During the boundary condition creation process, you will be required to select a wells data object from the **Data Explorer**.

A well can only be used if it meets the following requirements:

- The pumping well must be located within the simulation domain
- A screen must be defined for the pumping well
- A pumping schedule must be defined for the pumping well

For information on importing well data, please see “Importing Wells” on page 34. For information on defining well data for existing wells data objects, please see “Well Table” on page 65.

11.1.2 Specified Head

Currently, this boundary condition is only supported for Finite Difference Model translation.

The **Specified Head** boundary condition, also known as Constant Head in Visual MODFLOW, is used to fix the head value in selected grid cells regardless of the system conditions in the surrounding grid cells, thus acting as an infinite source of water entering the system, or as an infinite sink for water leaving the system. Therefore, specified head boundary conditions can have a significant influence on the results of a simulation, and may lead to unrealistic predictions, particularly when used in locations close to the area of interest.

During translation, Hydro GeoBuilder uses the **Time-Variant Specified-Head Package** provided with MODFLOW. The MODFLOW input data for Specified Head cells is stored in *projectname*.CHD file.

Unlike most other transient MODFLOW boundary condition packages, the Specified-Head package allows the specified heads to be linearly interpolated in time between the beginning and end of each stress period, such that the specified head for a grid cell may change at each time step of a given stress period.

Required Data

The Specified-Head package requires the following information for each specified head grid cell for each stress period:

- **Start Head:** Specified head value at the beginning of the stress period
- **Stop Head:** Specified head value at the end of the stress period

Supported Geometry

The geometry for Specified Head boundary conditions can be specified using the following data objects:

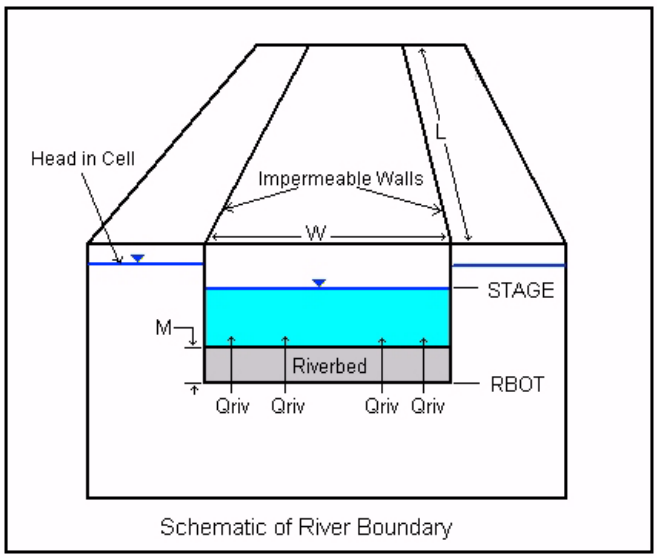
- Polyline
- Polygon

11.1.3 River

The River boundary condition is used to simulate the influence of a surface water body on the groundwater flow. Surface water bodies such as rivers, streams, lakes and swamps may either contribute water to the groundwater system, or act as groundwater discharge zones, depending on the hydraulic gradient between the surface water body and the groundwater system.

For finite difference models, Hydro GeoBuilder uses the **River Package** included with MODFLOW. The MODFLOW input data for River grid cells is stored in *projectname*.RIV file. Currently, translation of river boundary conditions is not supported for finite element (FEFLOW) translations.

The MODFLOW River Package simulates the surface water/groundwater interaction via a seepage layer separating the surface water body from the groundwater system (see following figure).



Schematic of River Boundary

Required Data

The MODFLOW River Package input file requires the following information for each grid cell containing a River boundary;

- **River Stage:** The free water surface elevation of the surface water body. This elevation may change with time.
- **Riverbed Bottom:** The elevation of the bottom of the seepage layer (bedding material) of the surface water body.
- **Leakance:** A numerical parameter representing the resistance to flow between the surface water body and the groundwater caused by the seepage layer (riverbed).

The Leakance value (C) may be calculated from the length of a reach (L) through a cell, the width of the river (W) in the cell, the thickness of the riverbed (M), and the vertical hydraulic conductivity of the riverbed material (K) using the following formula:

$$C = \frac{K \times L \times W}{M}$$

For situations where the River package is used to simulate lakes or wetlands, the L and W variables would correspond to the X-Y dimension of the River boundary grid cells.

When a River boundary condition is assigned, the **Use default Leakance** option is automatically selected.

If the **Use default Leakance** option is selected, the River boundary condition requires the following data:

- **River Stage**: The free water surface elevation of the surface water body.
- **Riverbed Bottom**: The elevation of the bottom of the seepage layer (bedding material) of the surface water body.
- **Riverbed Thickness**: Thickness of the riverbed (seepage layer).
- **Leakance**: A numerical parameter representing the resistance to flow between the surface water body and the aquifer (this field is read-only and is calculated using the formula described below).
- **Riverbed Kz**: Vertical hydraulic conductivity of the riverbed material.
- **River Width**: Width of the river.

When a polyline is used to define the river geometry, the default leakance formula is as follows:

$$\$COND = \frac{\$RCHLNG \times \$WIDTH \times \$K \times \$UCTOCOND}{\$RBTHICK}$$

When a polygon is used to define the river geometry, the default leakance formula is as follows::

$$\$COND = \frac{\$DX \times \$DY \times \$K \times \$UCTOCOND}{\$RBTHICK}$$

where

$COND: is the **Leakance**

$RCHLNG: is the reach length of the river line in each grid cell

$WIDTH: is the **River Width** in each grid cell

$K: is the **Riverbed Kz**

$UCTOCOND: is the conversion factor for converting the **$K** value to the same L and T units used by **$COND**

$RBTHICK:is the **Riverbed Thickness**

$DX: is the length of each grid cell in the X-direction

$DY: is the length of each grid cell in the Y-direction

If the **Use default Leakance** option is turned off, the fields used for calculating the River Leakance value (**Riverbed Thickness, Riverbed Kz**, and **River Width**) are removed from the table, and the **Leakance** field becomes a writable field where a value may be entered.

Supported Geometry

The geometry for River boundary conditions can be specified using the following data objects:

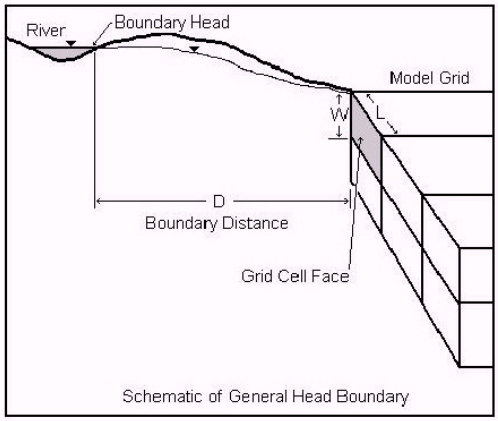
- Polyline
- Polygon

11.1.4 General Head

For finite difference models, Hydro GeoBuilder supports translation of the General-Head Boundary Package included with MODFLOW. The MODFLOW input data for General-Head grid cells is stored in the *projectname*.GHB file. Currently, for finite element models, translation of this boundary condition is not supported.

The function of the General-Head Boundary (GHB) Package is mathematically similar to that of the River, Drain, and Evapotranspiration Packages. Flow into or out of a cell from an external source is provided in proportion to the difference between the head in the cell and the reference head assigned to the external source. The application of this boundary condition is intended to be general, as indicated by its name, but the typical application of this boundary condition is to represent heads in a model that are

influenced by a large surface water body outside the model domain with a known water elevation. The purpose of using this boundary condition is to avoid unnecessarily extending the model domain outward to meet the element influencing the head in the model. As a result, the General Head boundary condition is usually assigned along the outside edges (sides) of the simulation model domain. This scenario is illustrated in the following figure.



The primary differences between the General-Head boundary and the Specified Head boundary are:

- the model solves for the head values in the General-Head grid cells whereas the head values are specified in Constant Head cells.
- the General-Head grid cells do not act as infinite sources of water whereas Specified Head cells can provide an infinite amount of water as required to maintain the specified head. Therefore, under some circumstances, the General-Head grid cells may become dry cells.

Required Data

The General-Head Boundary Package requires the following information for each General-Head grid cell:

- **Stage:** This is the head of the external source/sink. This head may be physically based, such as a large lake, or may be obtained through model calibration.
- **Leakance:** The leakance is a numerical parameter that represents the resistance to flow between the boundary head and the model domain.

In contrast to the River, Drain, and Evapotranspiration packages, the General Head package provides no limiting value of head to bind the linear function in either direction. Therefore, as the head difference between a model cell and the boundary head increases/decreases, flow into or out of the cell continues to increase without limit.

Accordingly, care must be used to ensure that unrealistic flows into or out of the system do not develop during the simulation.

The leakance value may be physically based, representing the conductance associated with an aquifer between the model area and a large lake, or may be obtained through model calibration. The leakance value (C) for the scenarios illustrated in the preceding figure may be calculated using the following formula:

$$C = \frac{(L \times W) \times K}{D}$$

where

(LxW) is the surface area of the grid cell face exchanging flow with the external source/sink

K is the average hydraulic conductivity of the aquifer material separating the external source/sink from the model grid

D is the distance from the external source/sink to the model grid

When a General-Head boundary condition is assigned, the **Use default leakance** option is automatically selected.

If the **Use default leakance** option is selected, the General-Head boundary condition requires the following data:

- **Stage:** The head value for the external source/sink
- **Leakance:** A numerical parameter representing the resistance to flow between the boundary head and the model domain (this field is read-only and is calculated using formula described below)
- **Distance to Reservoir:** The distance from the external source/sink to the General-Head grid cell
- **General Head Average Conductivity:** The average hydraulic conductivity of the aquifer material separating the external source/sink from the model grid

The default formula used to calculate the Leakance value for the General-Head boundary is:

$$\$COND = \frac{\$KAVG \times \$FACEAREA \times \$UCTOCOND}{\$DIST}$$

where

$COND: is the **Leakance** for each General-Head grid cell

$KAVG: is the **Average Conductivity**

$FACEAREA: is the surface area of the selected grid cell **Face** for each General-Head grid cell (automatically calculated during translation)

$UCTOCOND: is the conversion factor for converting the **$K** value to the same Length (L) and Time (T) units used by **$COND**

$DIST: is the **Boundary Distance,** the distance from the external source to the assigned general head boundary

If the **Use default conductance formula** option is not selected, the fields used for calculating the General-Head Conductance value (**Distance to Reservoir**, **Average Conductivity**) are removed from the table, and the **Leakance** field becomes a writable field where a value may be entered.

Supported Geometry

The geometry for General-Head boundary conditions can be specified using the following data objects:

- Polygon

11.1.5 Drain

For finite difference models, Hydro GeoBuilder supports the standard Drain Boundary Package included with MODFLOW. The MODFLOW input data for Drain grid cells is stored in the *projectname*.DRN file. Currently, for finite element model translation, this boundary condition is not supported.

MODFLOW's Drain Package is designed to simulate the effects of features such as agricultural drains, which remove water from the aquifer at a rate proportional to the difference between the head in the aquifer and some fixed head or elevation. The Drain package assumes the drain has no effect if the head in the aquifer falls below the fixed head of the drain.

Required Data

The Drain Package requires the following information as input for each cell containing this boundary condition:

- **Elevation:** The drain elevation, or drain head of the free surface of water within the drain. The drain is assumed to run only partially full, so that the head within the drain is approximately equal to the median elevation of the drain.
- **Leakance:** The drain leakance is a lumped coefficient describing the head loss between the drain and the groundwater system. This loss is caused by converging flow patterns near the drain, the presence of foreign material around the drain, channel bed materials, the drain wall, and the degree to which the drain pipe openings may be blocked by chemical precipitates, plant roots, etc.

There is no general formulation for calculating drain leakance. In most situations, the detailed information required to calculate drain leakance is not available to the groundwater modeler. These details include the detailed head distribution around the drain, aquifer hydraulic conductivity near the drain, distribution of fill material, number and size of the drain pipe openings, the amount of clogging materials, and the hydraulic conductivity of clogging materials. It is common to calculate drain leakance from measured values of flow rate and head difference. Drain leakance value is usually adjusted during model calibration.

When a polyline is used to define the boundary condition geometry, the default formula for the leakance is as follows:

$$\$COND = \$RCHLNG \times \$LCOND$$

When a polygon is used to define the boundary condition geometry, the default leakance formula is as follows:

$$\$COND = \$DX \times \$DY \times \$SCOND$$

where

$COND: is the **Leakance**

$RCHLNG: is the reach length of the drain in each grid cell

$LCOND: is the **Leakance per unit length** of the drain in each grid cell

$SCOND: is the **Leakance per unit area** of the drain in each grid cell

$DX: is the length of each grid cell in the X-direction

$DY: is the length of each grid cell in the Y-direction

If the **Use default leakance** option is turned off, the fields used for calculating the Drain Leakance value (**Leakance per unit length or area**) are removed from the table and the **Leakance** field becomes a read/write field where any value may be entered.

Supported Geometry

The geometry for General-Head boundary conditions can be specified using the following data objects:

- Polygon
- Polyline

11.1.6 Recharge

For finite difference models, Hydro GeoBuilder supports the Recharge Package (RCH) included with MODFLOW. The Recharge input data for MODFLOW is stored in the *projectname*.RCH file. For finite element models, recharge boundary conditions are translated as the In(+)/Out(-)flow material parameter.

The recharge boundary condition is typically used to simulate surficially distributed recharge to the groundwater system. Most commonly, recharge occurs as a result of precipitation percolating into the groundwater system. However, the recharge boundary can potentially be used to simulate recharge from sources other than precipitation, such as irrigation, artificial recharge, or seepage from a pond.

Note: The recharge rate is a parameter that is not often measured at a site, but rather, it is assumed to be a percentage of the precipitation. This percentage typically ranges from 5% to 20% depending on many different factors including:

- the predominant land use and vegetation type,
- the surface topography (slope), and
- the soil cover material

Supported Geometry

The geometry for Recharge boundary conditions can be specified using the following data objects:

- Polygon

11.1.7 Evapotranspiration

For finite difference models, Hydro GeoBuilder supports the Evapotranspiration Package (ET) included with MODFLOW. After translation, the Evapotranspiration input data for MODFLOW is stored in the *projectname*.EVT file. Currently, this boundary condition is not supported for finite element translation.

The evapotranspiration boundary condition simulates the effects of plant transpiration, direct evaporation, and seepage at the ground surface by removing water from the saturated groundwater regime.

The evapotranspiration boundary approach is based on the following assumptions:

- When the water table is at or above the ground surface (top of layer 1), evapotranspiration loss from the water table occurs at the maximum rate specified by the user.
- When the elevation of the water table is below the ‘extinction depth’, or is beneath layer 1, evapotranspiration from the water table is negligible.

Between these limits, evapotranspiration from the water table varies linearly with water table elevation.

Required Data

The Evapotranspiration Package requires the following information:

- **Evapotranspiration rate:** The rate of evapotranspiration as it occurs when the water table elevation is equal to the top of the grid cell elevation. This value should be entered in the units set for recharge as defined in the Project Settings.
- **Extinction Depth:** The depth below the top of grid cell elevation where the evapotranspiration rate is negligible.

Supported Geometry

The geometry for Evapotranspiration boundary conditions can be specified using the following data objects:

- Polygon

11.1.8 Lake

For finite difference models, Hydro GeoBuilder supports the Lake (LAK3) package for MODFLOW. After translation, the Lake input data for MODFLOW is stored in the *projectname*.LAK file. Currently, translation of this boundary condition is not supported for finite element models.

The lake boundary condition can be used to simulate the effects of stationary surface-water bodies such as lakes and reservoirs on an aquifer. The lake boundary is an alternative to the traditional approach of using the general head boundary condition. The main difference in the lake boundary is that the lake stage is calculated automatically based on the water budget, which is a function of inflow, outflow, recharge, etc.

For more information on the Lake package, please refer to USGS publication, *Documentation of a Computer Program to Simulate Lake-Aquifer Interaction Using the MODFLOW Ground-Water Flow Model and the MOC3D Solute-Transport Model*.

Required Data

The lake package requires the following input parameters:

- **Stage:** The initial stage of the lake at the beginning of the run.
- **Bottom:** The elevation of the bottom of the seepage layer (bedding material) of the surface water body.
- **Leakance:** A numerical parameter representing the resistance to flow between the boundary head and the model domain (this field is read-only and is calculated using formula described below)

- **Lakebed Thickness:** Thickness of the lakebed (seepage layer).
- **Lakebed Conductivity:** Vertical hydraulic conductivity of the lakebed material.
- **Precipitation Rate per Unit Area:** The rate of precipitation per unit area at the surface of the lake (L/T).
- **Evaporation Rate per Unit Area:** The rate of evaporation per unit area from the surface of the lake (L/T).
- **Overland Runoff:** Overland runoff (L^3/T) from an adjacent watershed entering the lake.
- **Artificial Withdrawal:** The volumetric rate, or flux (L^3/T) of water removal from a lake by means other than rainfall, evaporation, surface outflow, or ground-water seepage. Normally, this would be used to specify the rate of artificial withdrawal from a lake for human water use, or if negative, artificial augmentation of a lake volume for esthetic or recreational purposes.

The default leakance formula is as follows:

$$\$COND = \frac{\$DX \times \$DY \times \$K \times \$UCTOCOND}{\$RBTHICK}$$

where

$COND: is the **Leakance**

$K: is the **Lakebed Kz**

$UCTOCOND: is the conversion factor for converting the **$K** value to the same L and T units used by **$COND**

$RBTHICK:is the **Lakebed Thickness**

$DX: is the length of each grid cell in the X-direction

$DY: is the length of each grid cell in the Y-direction

If the **Use default Leakance** option is turned off, the fields used for calculating the River Conductance value (**Lakebed Thickness, Lakebed Kz**) are removed from the table, and the **Leakance** field becomes a writable field where a value may be entered.

Supported Geometry

The geometry for Lake boundary conditions can be specified using the following data objects:

- Polygon

11.1.9 Specified Flux

For finite difference models, Hydro GeoBuilder supports the Specified Flux (FHB1) package for MODFLOW. After translation, the specified flux input data for MODFLOW is stored in the *projectname*.FHB file. Currently, translation of this boundary condition is not supported for finite element models.

The Specified Flux boundary condition allows you to specify flow, as a function of time, at selected model cells. FHB1 is an alternative and (or) supplement to the recharge (RCH) package for simulating specified-flow boundary conditions. The main differences between the FHB1 package and the recharge package are as follows:

- FHB1 package can simulate specified-flux on the **top**, **side**, **bottom** or **intermediate** layers in the simulation domain, whereas the recharge package can only be applied to the **top** and **intermediate** layers.
- FHB1 package allows you to specify a starting flux and an ending flux (for each stress period, if transient). The package then uses linear interpolation to compute values of flow at each model time step.

For more information on the Specified-Flow (FHB1) package, please refer to *Documentation of a Computer Program (FHB1) for Assignment of Transient Specified-Flow and Specified-Head Boundaries in Applications of the Modular Finite-Difference Ground-Water Flow Model (MODFLOW,* Open-File Report 97-571, U.S. Geological Survey.

Required Data

The specified flux package requires the following input parameters:

- **Starting Flux (L^3/T)**
- **Ending Flux (L^3/T)**

Supported Geometry

The geometry for Specified Flux boundary conditions can be specified using the following data objects:

- Polygon
- Polyline

11.2 Defining a New Boundary Condition

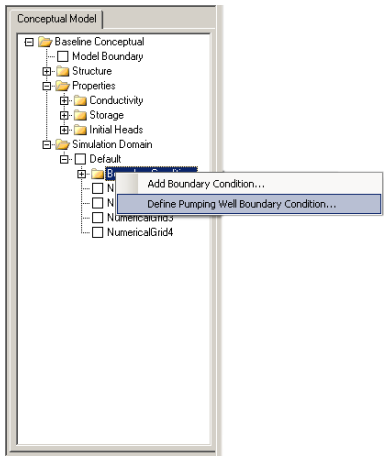
Note: Before you can define boundary conditions, you must first create the simulation model domain. For information on how to create the simulation model domain, please refer to “Creating the Simulation Model Domain” on page 139.

The workflow for defining a pumping well boundary conditions is different than that of defining other boundary conditions such as recharge, specified head, river, etc. Please see the following section for information on defining pumping well boundary conditions. For information on how to define all other types of boundary conditions, please skip to “Defining Other Boundary Conditions” on page 158.

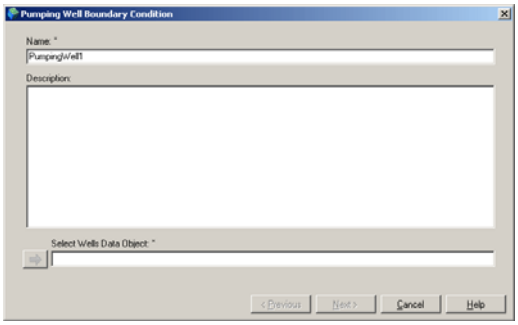
11.2.1 Defining a Pumping Well Boundary Condition

To add a new pumping well boundary condition, follow the steps below:

- From the **Conceptual Model** tree, right-click on **Boundary Conditions** and select **Define Pumping Well Boundary Condition** from the pop-up menu.

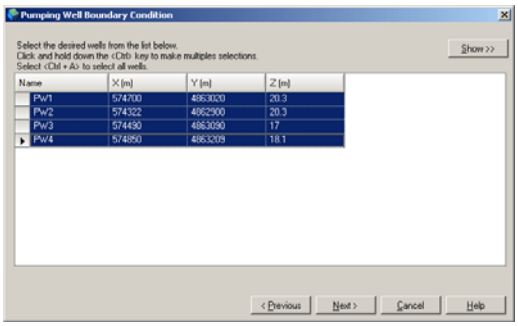


The **Pumping Well Boundary Condition** dialog will display on your screen (shown below).



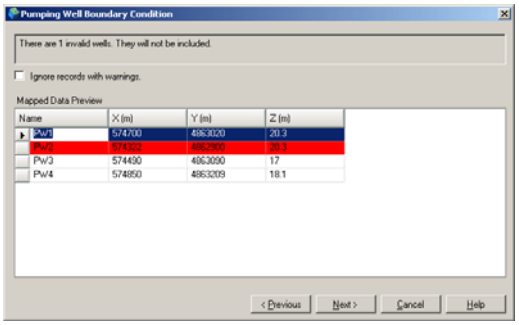
- Type in a unique name for the pumping well boundary condition in the **Name** field. This name will appear in the **Conceptual Model** tree, under **Boundary Condition** node, when the boundary condition is created.
- Type in a description of the boundary condition in the **Description** field *(optional)*.
- Select a pumping wells data object from the **Data Explorer**, and then click the button to insert the data object into the **Select Wells Data Object** field.
- Click the **[Next]** button to proceed to the next step.

The next step involves selecting which wells to include in the pumping well boundary condition.



- Highlight the desired wells from the list. Press the **[CTRL]** key on your keyboard to make multiple selections, or press **[CTRL] + A** to select all the wells in the table.
- Click the **[Show]** button to preview the selected wells in a 3D Viewer.
- Click the **[Next]** button to proceed to the next step.

In the next dialog (shown below), Hydro GeoBuilder validates the selected pumping well data and highlights any wells that do not meet the data requirements.



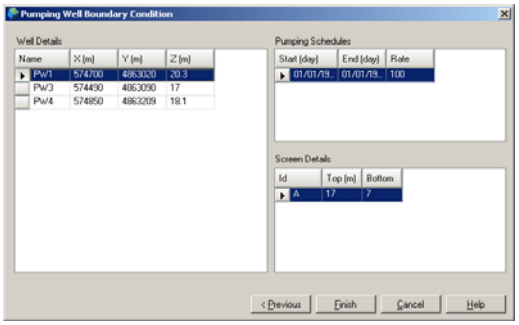
Each pumping well must satisfy the following requirements in order to be deemed valid:

- The pumping well must located within the simulation domain.
- A screen must be defined for the pumping well
- A pumping schedule must be defined for the pumping.

Wells that do not meet these requirements will be highlighted red in the **Mapped Data Preview** table, and cannot be used in defining the pumping well boundary condition. For information on defining well data, *i.e.*, screens, pumping schedules, please see “Well Table” on page 65.

To omit any invalid wells from the boundary condition, simply click the **Ignore Records with Warnings** check box, and then click the **[Next]** button to continue.

The next (and last) dialog allows you to preview the pumping well data before creating the pumping well boundary condition.



- Select a well from the **Well Details** table, and the corresponding pumping schedule and screen details will appear in the adjacent tables.

Click the **[Finish]** button to finalize the pumping well boundary condition. Hydro GeoBuilder will then add the boundary condition under the **Boundary Condition** node in the **Conceptual Model** tree.

11.2.2 Defining Other Boundary Conditions

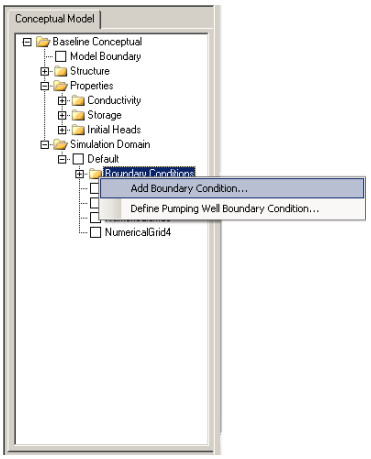
This section describes the workflow for creating the following boundary conditions:

- Specified Head
- River
- General Head
- Drain
- Recharge
- Evapotranspiration
- Lake
- Specified Flux

Note: For information on creating a Well boundary condition, please see “Defining a Pumping Well Boundary Condition” on page 155

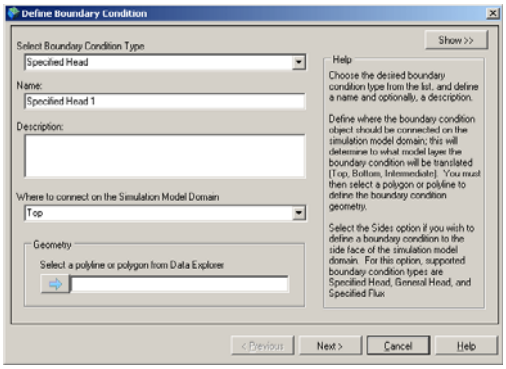
To create a boundary condition, follow the steps below:

- From the **Conceptual Model** tree, right-click on the **Boundary Condition Container** node and select **Add New Boundary Condition...** .



Specifying Boundary Condition Type and Location

The first step involves selecting the boundary condition type and specifying the location of the boundary condition on the simulation domain.


Define Boundary Condition
Select Boundary Condition Type
Specified Head
Name:
Specified Head 1
Description:
Where to connect on the Simulation Model Domain
Top
Geometry
Select a polyline or polygon from Data Explorer
Show >>
Help
Choose the desired boundary condition type from the list, and define a name and optionally, a description.
Define where the boundary condition object should be connected on the simulation model domain; this will determine to what model layer the boundary condition will be translated (Top, Bottom, Intermediate). You must then select a polygon or polyline to define the boundary condition geometry.
Select the Sides option if you wish to define a boundary condition to the side face of the simulation model domain. For this option, supported boundary condition types are Specified Head, General Head, and Specified Flux
< Previous
Next >
Cancel
Help

- From the **Select Boundary Condition Type** combo box, select the desired boundary condition type. For more information on each boundary condition type including the data requirements for MODFLOW, please see “Boundary Conditions Overview” on page 142.
- Enter a **Name** and a **Description** (optional) for the boundary condition. The specified name will appear in the **Conceptual Model** tree, once the boundary condition is created.
- Next, select where to apply the boundary condition on the simulation domain by selecting an option from the **Where to apply on the Simulation Model Domain** combo box. The type of options available in this combo box depend on which boundary condition type is selected. The table below summarizes the available options for each boundary condition type:

	Top	Bottom	Side	Intermediate
Specified Head	✔	✔	✔	✔
River	✔	✔		✔
General Head	✔	✔	✔	✔
Drain	✔	✔		✔
Recharge	✔			
Evapotrans.	✔			
Lake	✔			✔
Specified Flux	✔	✔	✔	✔

Each option is described below:

Top

Selecting this option will apply the boundary condition to the top layer of the simulation domain.

Bottom

Selecting this option will apply the boundary condition to the bottom layer of the simulation domain.

Side

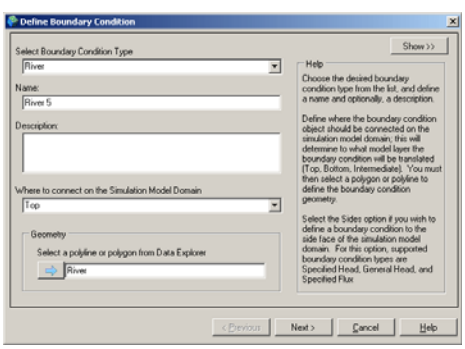
Selecting this option will allow you to apply the boundary condition to a single side or combination of sides of the simulation domain. When this option is selected, you will be required to define the geometry of the boundary condition by selecting the desired sides of the simulation domain using an interactive 3D Viewer. For more information on how to do this, please see “Defining Boundary Condition Geometry (Sides)” on page 163

Intermediate

Selecting this option will allow you to assign the boundary condition to an intermediate model layer within the simulation domain, by specifying a **Connection Elevation** value. Upon translation, Hydro GeoBuilder will assign the boundary condition to the appropriate model layer based on the defined connection elevation value.

Specifying Boundary Condition Geometry (Horizontal)

The next step is to define the geometry of the boundary condition. Boundary conditions that are applied to the *top*, *bottom* or *intermediate layers* can be defined by using a **Polygon** (areal) or **Polyline** (linear) data object from the **Data Explorer**.


Define Boundary Condition
Show >>
Select Boundary Condition Type
River
Name:
River 5
Description:
Where to connect on the Simulation Model Domain
Top
Geometry
Select a polyline or polygon from Data Explorer
River
Help
Choose the desired boundary condition type from the list, and define a name and optionally, a description.
Define where the boundary condition object should be connected on the simulation model domain; this will determine to what model layer the boundary condition will be translated (Top, Bottom, Intermediate). You must then select a polygon or polyline to define the boundary condition geometry.
Select the Sides option if you wish to define a boundary condition to the side face of the simulation model domain. For this option, supported boundary condition types are Specified Head, General Head, and Specified Flux.
< Previous
Next >
Cancel
Help

- From the **Data Explorer**, simply the polygon or polyline data object that

represents the geometry of the boundary condition, and then click the button to insert the data object into the **Select a polyline or polygon from the Data Explorer** field.

Note: Some boundary conditions only support one type of geometry, either polygon or polyline.

Polygons and polyline data objects that extend beyond the model domain must be "clipped" before they can be used to define boundary conditions. Polygons and polylines can be clipped using the "clip to polygon" data operations, which can be accessed in the data object settings (right-click on the polygon or polyline in the **Data Explorer**, and select **Settings...** from the pop-up menu). For more information on data object operations, please see "Performing Operations on Data" on page 70.

Click the **[Next]** button to proceed to the next step.

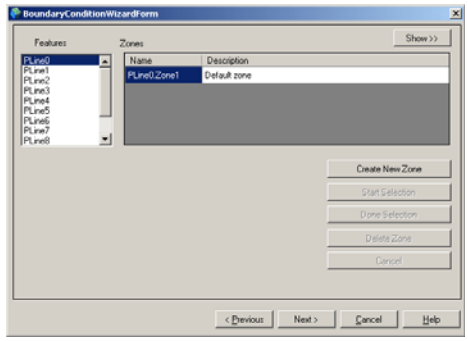
For boundary conditions that are to be applied to the **Side(s)** of the simulation domain, Hydro GeoBuilder allows you to define the geometry by selecting the appropriate side(s) using an interactive 3D Viewer window. This process is described in "Defining Boundary Condition Geometry (Sides)" on page 163.

If you select a **polygon** data object to define the geometry of the boundary condition, the next step is defining boundary condition parameters. Please skip to "Defining Boundary Condition Parameters" on page 165 for information on this topic.

If you select a **polyline** data object to define the geometry of the boundary condition, the next step is defining zones for the selected polyline. This procedure is described in the following section.

Defining Polyline Zones

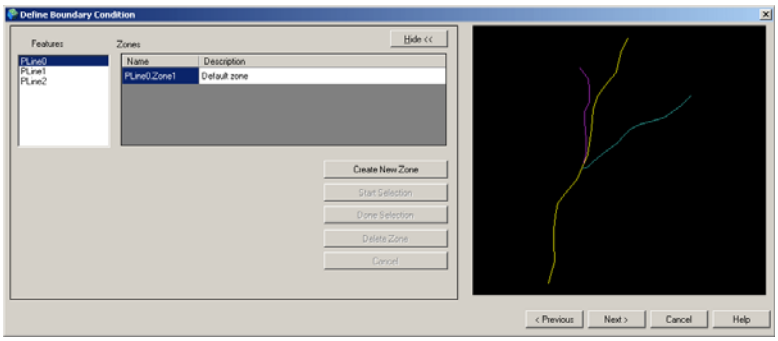
When a polyline data object is selected for defining the geometry of the boundary condition, Hydro GeoBuilder automatically creates a new zone for each individual line in the polyline data object (polyline data objects can consist of multiple lines).



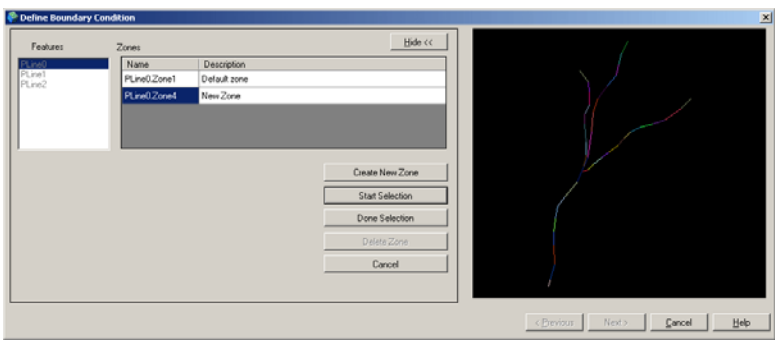
The lines that comprise the selected polyline data object are listed in the **Features** list box, *e.g.*, PLine0, Pline2, Pline3 etc. When a line is selected, its corresponding zones are shown in the adjacent **Zones** table. Each line has a default zone, which represents the entire length of the line. However, if the default zones are not suitable, you can create new zones. This is accomplished by selecting single or multiple line segments from polylines, using an interactive 2D Viewer window.

To create a new zone, follow the steps below:

- Click the **[Show]** button to show the interactive 2D Viewer window.

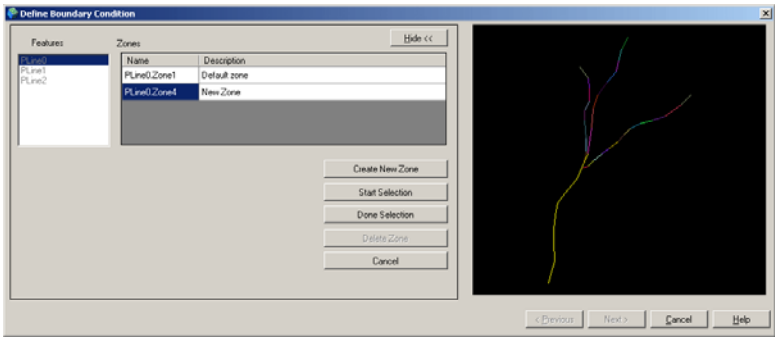


- Select the desired polyline from the **Features** list. The corresponding line will be highlighted yellow in the adjacent 2D Viewer.
- Click the **[Create New Zone]** button. A new row will be added to the **Zones** table where you can enter a **Name** and a **Description** for the new zone (optional).
- Click the **[Start Selection]** button. Once selected, the line segments that make up the polyline will be visible and shown in different colors in the 2D Viewer (shown below)



- While holding the **[CTRL]** button on your keyboard, click the desired line segments that will represent the new zone, from the interactive 2D Viewer. When selected, the line segments will be highlighted yellow. **Note:** You can

only select the line segments from the line selected in the **Features** list.

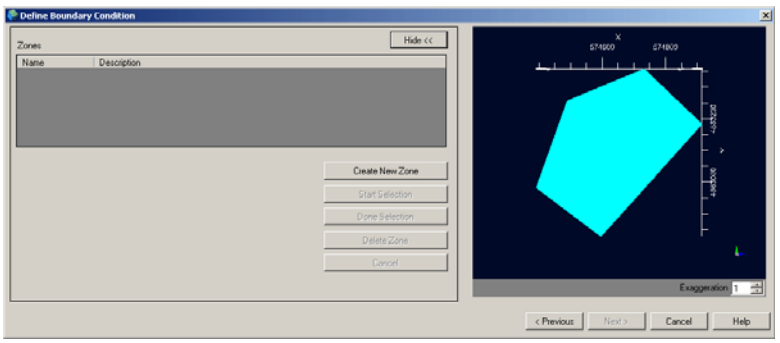


- Once you have selected the desired line segments, click the **[Done Selection]** button to save the selection.
- Repeat the steps above for creating additional zones.

To delete a zone, simply select the zone from the **Zones** table, and then click the **[Delete Zone]** button.

Defining Boundary Condition Geometry (Sides)

If you choose to apply the boundary condition to the **Sides** of the simulation domain, click the **[Next]** button to select which side(s) to apply the boundary condition to.



Applying boundary conditions to the sides of the simulation domain is accomplished by manually selecting the desired sides using an interactive 3D Viewer window.

- Click the **[Show]** button to display the interactive **3D Viewer**.

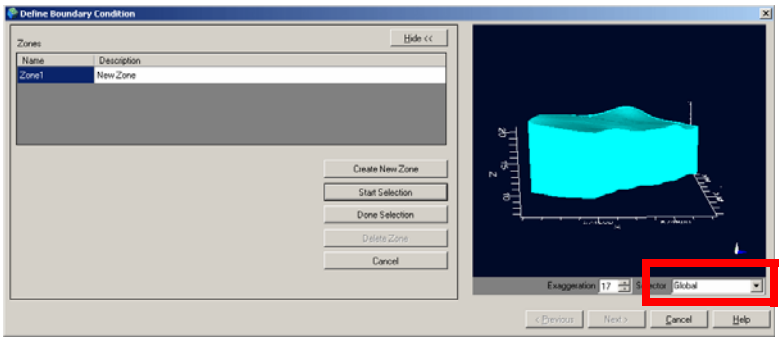
The interactive 3D Viewer behaves just like any other 3D Viewer in Hydro GeoBuilder. You can zoom in and out, rotate and move the displayed simulation domain using your

mouse. You can also change the color of the background, show/hide the axis, and change the vertical exaggeration.

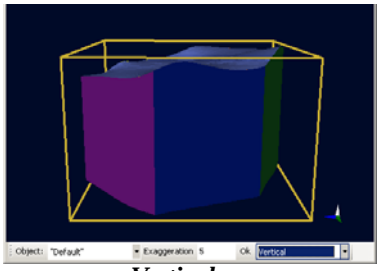
Creating a New Zone

To create a new zone, follow the steps below:

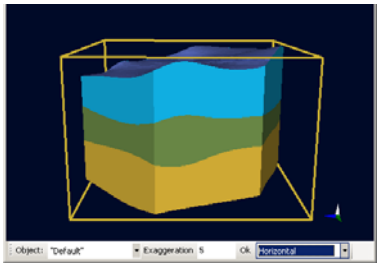
- Click the **[Create New Zone]** button. A new row will be added to the Zones table. Here you can change the zone **Name** and **Description**, as desired.
- Click the **[Start Selection]** button. A new combo box called **Selector** will be added to the bottom of the interactive 3D Viewer (indicated below).



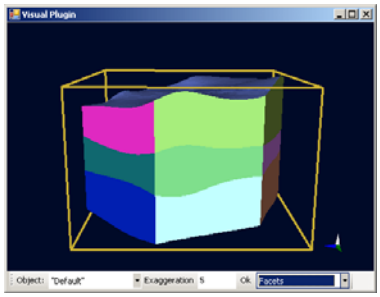
- From the **Selector** combo box, select one of the following options:
  - **Global:** Select all sides around the entire simulation domain
  - **Horizontal**: Select the area(s) between two horizons, around the entire simulation domain.
  - **Vertical**: Select an area(s) between two edges, spanning the entire height of the simulation domain.
  - **Facets**: Select the area(s) that are bounded on the sides by edges of the simulation domain, and bounded at the top/bottom by horizons.



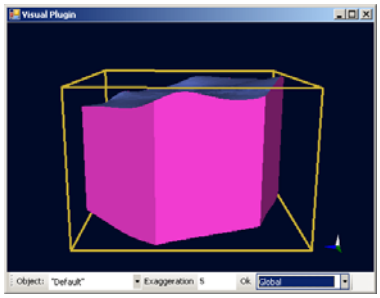
Vertical



Horizontal



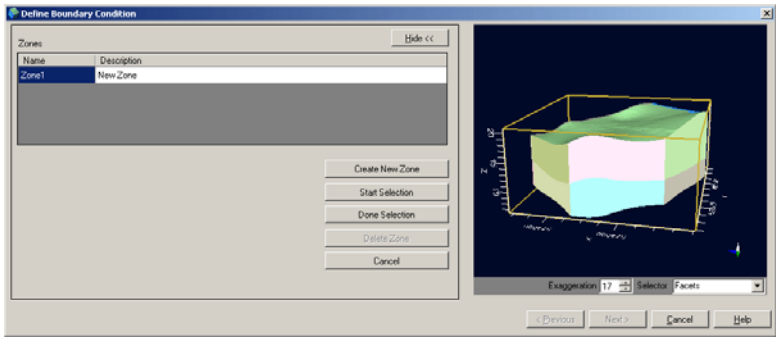
Facets



Global

- Using your mouse, click on the simulation domain in the 3D Viewer, and select the appropriate sides for the boundary condition. When a side is selected, it will become highlighted.
- **Tip!** You can select multiple sides by holding down the **[CTRL]** on your keyboard

.



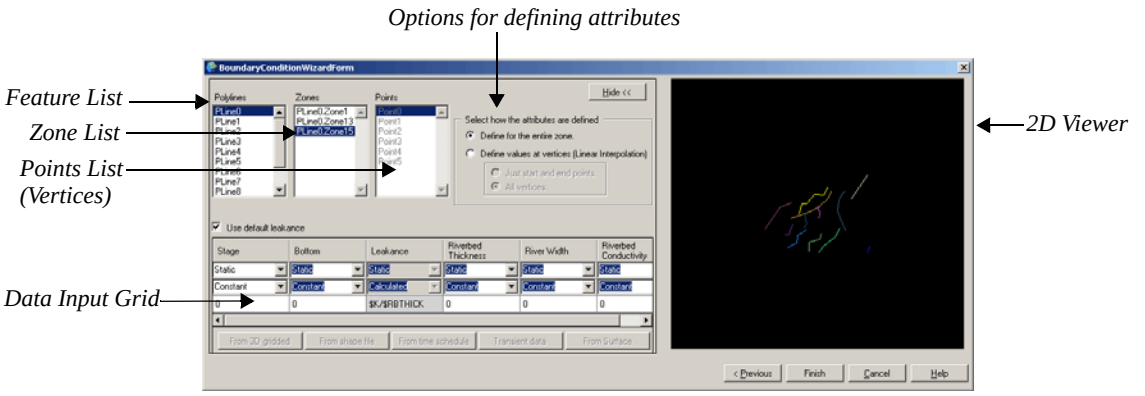
- Once the sides have been highlighted, click the **[Done Selection]** button to save the selections.
- Repeat the steps above to create additional zones.

Defining Boundary Condition Parameters

Note: Parameter values must be entered in the units defined in the project settings. To read more about the project settings, please see “Units” on page 18.

Once the geometry has been defined (see previous sections), the next step is to define the boundary condition parameters. Although each boundary condition type requires a different set of parameters, the data input windows each have similar features and functionality. For information on required parameters for each boundary condition, please see the appropriate heading under “Boundary Conditions Overview” on page 142.

A typical boundary condition data input window is shown below:



Select the Method for Defining Attributes (Polylines Only)

For polylines, there are two ways in which you can assign attributes to the boundary condition geometry:

- **Define for the entire zone (default):** This option allows you to assign boundary condition data to the entire zone.
- **Define values at vertices :** This option allows you to assign boundary condition data to the vertices along the zone (line), and then during translation, linear interpolation is used to determine the parameters for the cells that fall between the specified vertices. With this method, there are two options:
    - **Define Start and End Points:** This option allows you to define the attribute value only at the *first* and *last* vertex of a zone.
    - **All Vertices:** This option allows you to define attribute values for each vertex in a zone (first, last and all intermediate vertices).

Linear Interpolation Explained

When the **Define values at vertices** option is selected, two sets of boundary condition data are required; one set for the grid cell at the **Start Point** of the line (or line segment), and one set for the grid cell at the **End Point** of the line (or line segment). The boundary condition data for the grid cells between the **Start Point** grid cell and the **End Point** grid cell will be linearly interpolated between these two points using the formula below:

$$\$X_i = \$X_{SP} + (\$X_{SP} - \$X_{EP}) \times \frac{\$TVAR_{1,i}}{\$LENGTH}$$

where

$X_i is the boundary condition parameter value at the i^{th} grid cell along the line

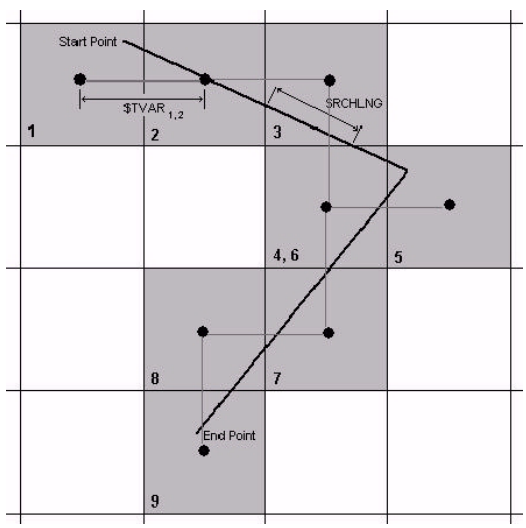
$X_{SP} is the boundary condition parameter value at the **Start Point** of the line

$X_{EP} is the boundary condition parameter value at the **End Point** of the line

$TVAR_{1,i} is the cumulative length of the line at the i^{th} grid cell along the line, as measured from the center of the **Start Point** grid cell through the center of each successive grid cell along the line (see following figure).

$LENGTH is the total length of the line, as measured from the center of the **Start Point** grid cell through to the center of the **End Point** grid cell (see following figure).

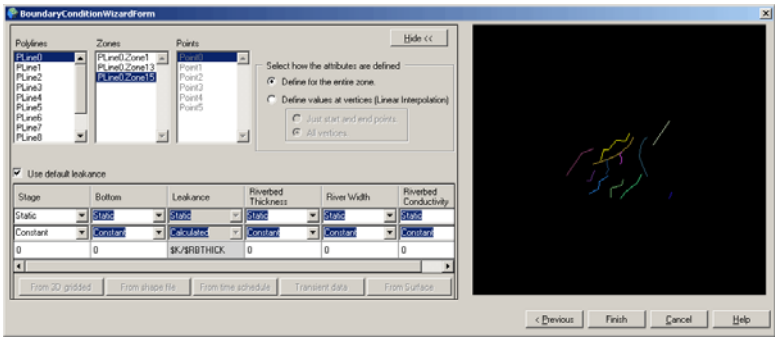
When the line is digitized from the **Start Point** to the **End Point**, each grid cell is numbered in sequence according to the order in which the line passes through each cell. If the line passes through the same grid cell twice, the grid cell will be numbered twice as seen for grid cell “4” and “6” in the following figure. As a result, the parameter value calculated for “grid cell #6” will over-write the parameter value calculated for “grid cell #4”.



Specifying Boundary Condition Data

Boundary condition data can be specified by using the **Data Input Grid**. Each column in the data input grid represents a required attribute for the selected boundary condition.

For more information on the required parameters for each boundary condition, please see the appropriate section under “Boundary Conditions Overview” on page 142.



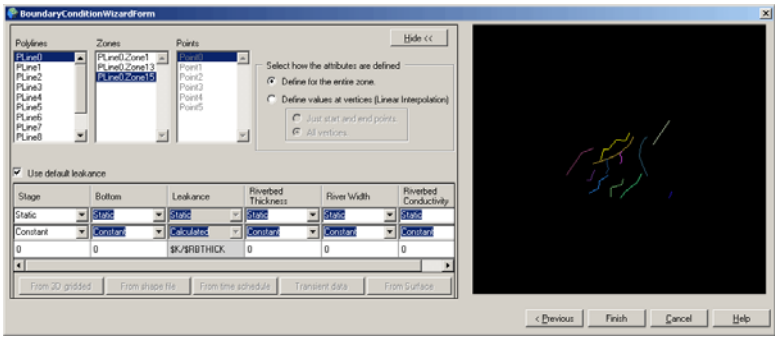
Attribute data can be defined for each zone in the boundary condition geometry (polygon or polyline). Simply select the feature for which attributes are to be defined from the **Feature List**, select a zone from the **Zone** list, and then define the attribute data in the **Data Entry Grid**. Repeat this process for other features/zones in the selected data object.

Note: The selected zone will be highlighted yellow in the adjacent 2D Viewer preview.

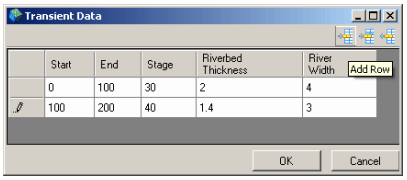
If attribute data is not defined for certain zones, these parts of the boundary condition will not be included during translation.

For polylines only, it is possible to define attributes at line vertices. In this case, the **Points List** will become available, where you can select the vertices that comprise the selected zone. For more information on assigning attributes to points, see “Select the Method for Defining Attributes (Polylines Only)” on page 166.

For each attribute in the **Data Input Grid**, there are two combo boxes.



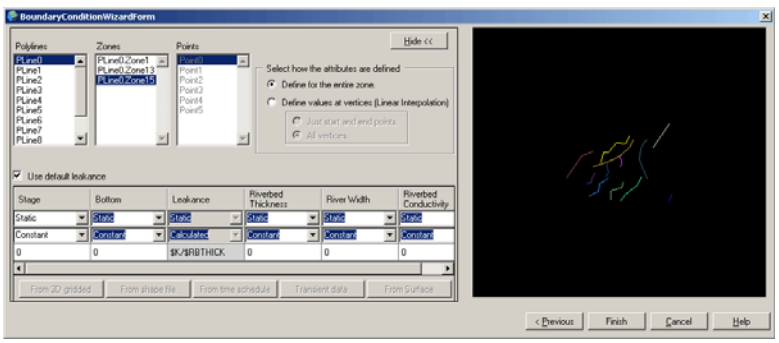
The first combo box allows you to set an attribute as **Static** (**Steady-State)** or **Transient** (conditions change over time). When one or more parameters are set to **Transient**, the **Transient Data** button will become active. When selected, the **Transient Data** window will launch (shown below).



The **Transient Data** dialog allows you to define the stress periods and values for all the attributes in the boundary condition that have been set as “Transient”.

- Click the **Add Row** button to add a new row to the table.
- Enter a **Start** and **End** time, and a **Value** for each transient attribute.
- Press the **[Enter]** button on your keyboard.
- Repeat for additional stress periods.
- Click **[OK]** to save the transient data.

The second combo box provides different methods for assigning attribute values to the boundary condition.



The contents shown in this combo box depend on the attribute type, *i.e.*, not all methods are available for every attribute. The available methods may include: **Constant Value**, **From 3D Gridded Data**, **From Shapefile, From Time Schedule** and **From Surface**. Each method is described below:

Constant Value

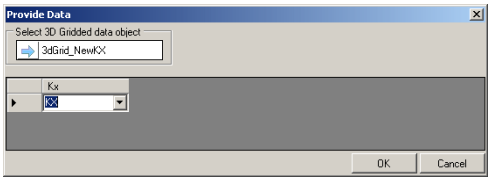
The constant value method allows you to define a single value for the entire zone. Upon translation, each grid cell comprising the boundary condition zone will be assigned the specified constant value.

When this method is selected (default), simply enter the desired attribute value in the **Data Entry Grid**.

Note: The values for each constant value attribute should be entered in the same units as defined in the **Project Settings**.

From 3D Gridded Data

This method allows you to use spatially-variable attributes from a 3D Gridded data object for defining a boundary condition attribute. When this method is selected, the **Use 3D Gridded Data** button will become active. When selected, the 3D Gridded Data dialog will launch (shown below).

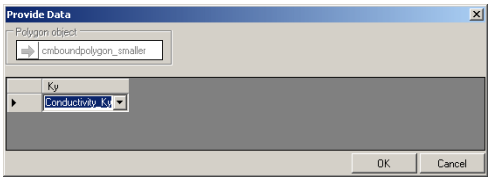


From the **Data Explorer**, select the desired 3D Gridded data object and then click the button to insert it into the **Select 3D Gridded Data Object** field. Once selected, the data object's attributes are listed in the combo box below. Select the desired attribute from the combo box, and then click the **[OK]** button to close the dialog box.

Note: The specified 3D Gridded data object must horizontally and vertically overlap the defined property zone geometry, or else the data object cannot be used.

From Shapefile

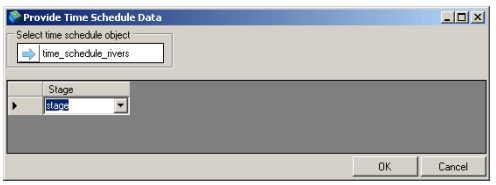
The method allows you to use **Shapefile** attributes for defining boundary condition attributes. When this method is selected, click the **Use Shapefile** button to launch the **Shapefile** dialog (shown below):



The combo box contains all the attributes of the specified polygon used to define the horizontal geometry of the boundary condition. Select the desired attribute from the combo box, and then click the **[OK]** button to close the dialog box.

From Time-Schedule (Transient only)

This method allows you to use a time-schedule data object for defining the stress periods and values of a transient attribute. When this method is selected, click the **Use Time Schedule** button to launch the **Time Schedule** dialog.



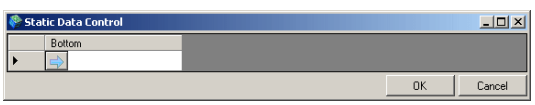
From the **Data Explorer**, select the desired time schedule data object and then click the button to insert it into the **Select Time Schedule Object** field. Once selected, the data objects attributes are listed in the combo box below. Select the desired attribute value from the combo box, and then click the **[OK]** button to close the dialog box.

From Surface

This method allows you to define boundary condition attributes using an existing **Surface** data object. Upon translation, attribute values are calculated from the specified surface data object. A surface data object can be useful for defining an elevation attribute, *i.e.*, River Stage, Head, Lakebed Bottom etc.

Note: The surface data object must cover the entire conceptual model domain area.

When this method is selected, click the **[From Surface]** button to launch the **Static Data Control** dialog (shown below).



To specify a surface data object,

- Select the desired surface from the **Data Explorer**.
- Click the button to insert the surface data object into the attribute field.
- Repeat for other attributes that have been assigned this method.
- Click the **[OK]** button.

Use Default Leakance

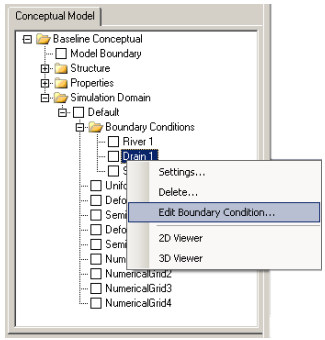
The **Use Default Leakance** option is used to calculated the leakance value for River, Drain, Lake and General Head boundary conditions using a mathematical expression containing array variables (see the section “Using Mathematical Formulas and Array Variables in the Visual MODFLOW User’s Manual for more information). If the Use Default Leakance option is selected, the leakance value will be calculated using a default formula associated with each boundary condition type. If this option is not selected, a leakance value will need to be entered manually.

The advantage of using the default leakance formula to calculate the leakance value for the group of grid cells is that each grid cell will be assigned a leakance value proportional to the size of the grid cell.

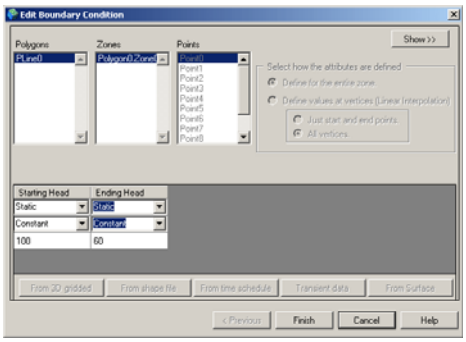
11.3 Modifying Boundary Conditions

To view and modify parameter attributes for existing boundary conditions, follow the steps below:

- From the **Conceptual Model** tree, right-click on the desired boundary condition, and select **Edit Boundary Condition...** from the pop-up menu.



- The **Edit Boundary Condition** dialog box will display on your screen, allowing you modify the input parameters for the boundary condition. For more information on defining parameter attributes, please refer to “Defining Boundary Condition Parameters” on page 165.



- Once modifications have been made to the input parameters, click the **[Finish]** button to save the changes.

11.4 Deleting Boundary Conditions

To delete a boundary condition, follow the steps below:

- From the **Conceptual Model** tree, right-click on the desired boundary condition and select **Delete** from the pop-up menu.
- You will prompted with a confirmation message. Click the **[Yes]** button to delete the boundary condition.

Note: Please be aware that there is no undo function to recover a deleted boundary condition. Please exercise caution when deleting boundary conditions.

12

Model Domain Discretization

Once you have created your conceptual model, and have generated the simulation domain, you can discretize your model using either the finite difference method or the finite element method. The finite difference method involves fitting your conceptual model to a finite difference grid. Once translated, your model can then be opened and simulated in Visual MODFLOW. The finite element method involves fitting your conceptual model to a finite element mesh. Once translated, your model can be opened and simulated using FEFLOW.

This chapter contains information on the following topics:

Creating a Finite Difference Grid

- Creating a Finite Difference Grid
- Defining the Horizontal Grid
- Defining the Vertical Grid
- Editing a Finite Difference Grid
- Deleting a Numerical Grid
- Horizontal Grid Refinement
- Grid Types
- Creating a Child Grid

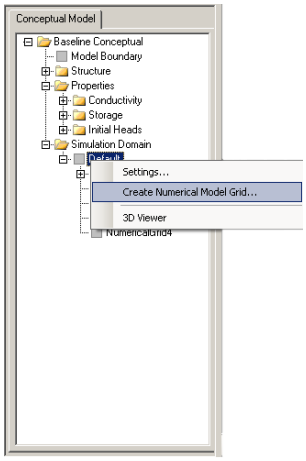
Creating a Finite Element Mesh

- Defining the Superelement Mesh
- Defining the Horizontal Mesh Settings
- Delaunay Triangulation Method (L-switch)
- Polygon Refinement
- Defining Slice Elevations

12.1 Creating a Finite Difference Grid

To create a numerical model grid, follow the steps below

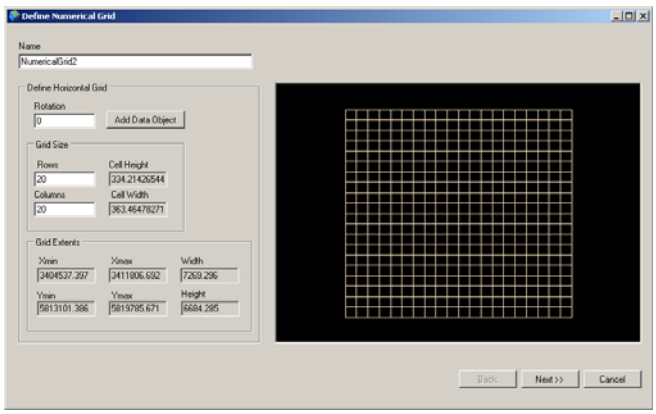
- From the **Conceptual Model** tree, right-click on the **Simulation Domain** node for which the grid is to be created, and select **Create Numerical Model Grid...**.



- The next step involves defining the horizontal grid discretization for the simulation domain, and is described in the following section.

12.1.1 Defining the Horizontal Grid

By default, Hydro GeoBuilder discretizes the horizontal grid using 20 rows and 20 columns, with no rotation. However, you can customize the grid to your liking, by modifying the settings in the horizontal grid dialog (shown below).



Enter a unique **Name** for the numerical grid. This name will appear in the **Conceptual Model** tree once the grid is created.

The grid can be rotated counter-clockwise about the grid origin by entering a value between 0 and 360 in the **Rotation** text field.

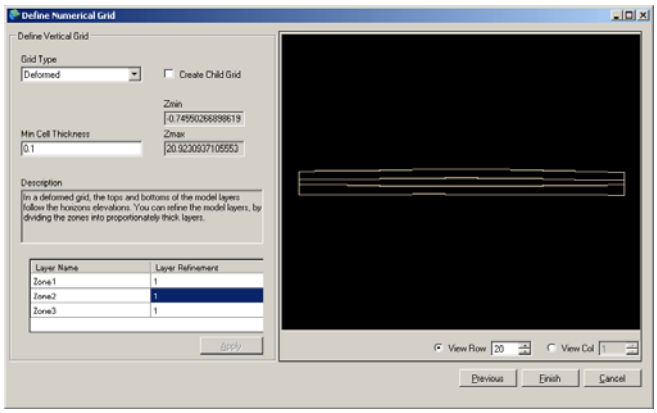
The **Add Data Object** button allows you to display data objects, *e.g.*, model features, in the adjacent 2D Viewer. Adding model features to the grid designer may help you determine the appropriate dimensions and rotation for the numerical grid. Simply select a data object from the **Data Explorer** and then click the **Add Data Object** button to show the data object in the 2D Viewer.

The **Xmin** and **Ymin** values refer to the X-Y coordinates of the bottom-left corner of the numerical grid. The **Xmax** and **Ymax** values refer to the X-Y coordinates of the top-right corner of the numerical grid.

The **Columns** and **Rows** fields allow you to define the **Grid Size**. The maximum grid size supported by Hydro GeoBuilder is 5000 rows by 5000 columns. **NOTE:** If you intend on importing the numerical model into Visual MODFLOW, the grid size cannot be larger than 500 grid lines (499 columns) in the X-direction by 500 grid lines (499 rows) in the Y-Direction by 200 layers in the Z-direction. If these dimensions are insufficient for a specific problem, please contact SWS for a customized version of Visual MODFLOW compiled specifically for the grid dimensions required.

Click the **[Next]** button to proceed to define the vertical discretization.

12.1.2 Defining the Vertical Grid

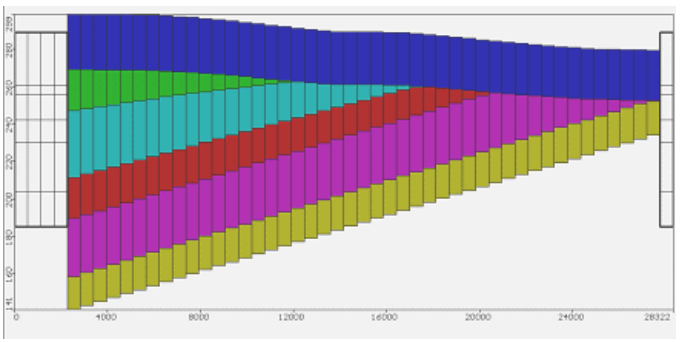


The first step in defining the vertical grid is selecting the **Grid Type**. There are three different grid types: **Deformed**, **Uniform** and **Deformed-Uniform**. Each grid type is described in the following sections.

Grid Types

Deformed

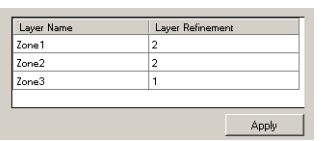
In a deformed grid, the tops and bottoms of the model layers conform to the horizons elevation. You can refine the model layers, by diving the structural zones into proportionately thick layers.



Cross sectional view of deformed grid from Visual MODFLOW

A **Minimum Cell Thickness** must be specified as MODFLOW does not permit lateral discontinuity of layers, *i.e.*, a layer cannot have a thickness of 0 at any point in the layer. When horizons are on-lapping one another, resulting in a zero cell thickness, the minimum cell thickness is applied and the horizons are shifted based on the horizon types defined in the Horizon settings (See “Horizon Types” on page 121).

For deformed grids, you have the option of refining (subdividing) each layer into a specified number of equally thick layers. In the table located below the grid description, enter a refinement factor for the desired layer(s). For example, a layer refinement factor of 2 would subdivide the layer into two equally spaced layers.

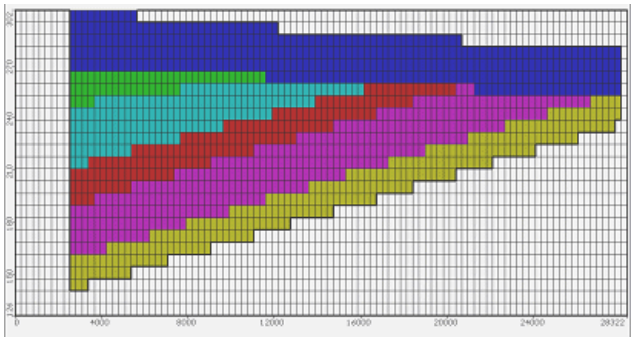


After entering a refinement factor, click the **[Apply]** button to view the changes in the adjacent 2D Viewer.

Uniform

In a uniform grid, a number of layers with uniform thickness will be created. At the time of translating the conceptual model to the numerical model, the properties will be assigned to the appropriate grid cells to represent the geological structure. This grid is

useful for transport or density-dependent simulations, where it is desirable to have fine vertical discretization.



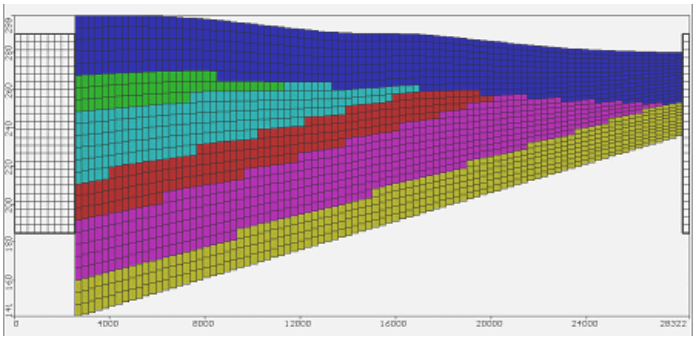
Cross sectional view of uniform grid from Visual MODFLOW

When this grid type is selected, specify the number of layers to create in the **Number of Layers** field (default is 10).

Note: Hydro GeoBuilder only supports up to 1000 layers in a numerical grid.

Deformed-Uniform

In a deformed-uniform grid, the top and bottom of the grid are deformed, following the top-most and bottom-most horizons respectively; in between, a set of uniformly thick layers will be generated. At the time of translating the conceptual model to the numerical model, the properties will be assigned to the appropriate grid cells to represent the geological structure. This grid is useful where you have discontinuous layers.



Cross sectional view of deformed-uniform grid from Visual MODFLOW

For Deformed-Uniform grids, you must specify a **Minimum Cell Thickness** (see above) and the **Number of Layers**.

Creating a Child Grid

A higher-resolution block-shaped child grid can be created within any numerical grid (parent grid) and used for running Local Grid Refinement (LGR) simulations with the MODFLOW-2005 LGR package. Refined child grids are often used to improve simulation accuracy around areas of interest within your simulation domain. For example, refined grids may be needed in:

- regions where hydraulic gradients change substantially over short distances, as would be common near pumping or injecting wells, rivers, drains, and focused recharge.
- regions of site-scale contamination within a regional aquifer where simulations of plume movement are of interest.
- regions requiring detailed representation of heterogeneity, as may be required to simulate faults, lithologic displacements caused by faulting, fractures, thin lenses, pinch outs of geologic units, and so on.

Hydro GeoBuilder allows you to create up to 9 child grids within a single parent grid. However, you cannot create a child grid within a child grid, and child grids cannot not overlap within a single parent grid.

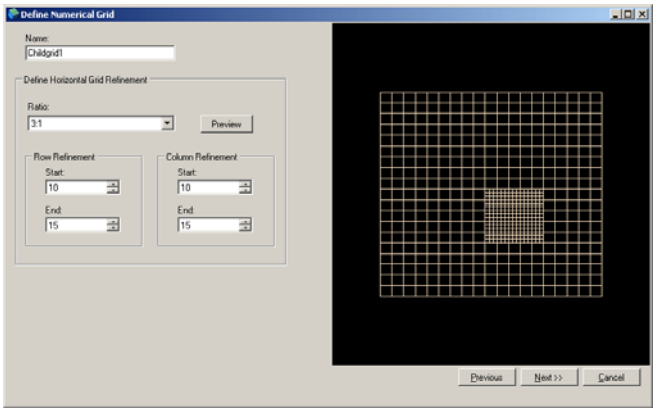
For more information on the local grid refinement package, please refer to *MODFLOW-2005, The U.S Geological Survey Modular Ground-Water Model-Documentation of Shared Node Local Grid Refinement (LGR) and the Boundary Flow and Head (BFH) Package*, by Steffen W. Mehl and Mary C. Hill, U.S. Geological Survey.

There are two ways in which you can define a child grid within a numerical model grid.

- When defining the parent grid; check the **Create Child Grid** box in the **Vertical Grid** dialog, and click the **[Next]** button.
- After the parent grid has been defined; right-click on the numerical grid from the **Conceptual Model tree**, and select **Create Child Grid**.

Horizontal Grid Refinement

Horizontal child grid refinement involves specifying the location of the child grid within the parent grid, and defining the row and column refinement ratio.



Tip! You can add data objects (e.g., boundaries, wells, site maps) from the data explorer to the **2D Viewer** preview to assist you in determining the placement of the child grid within the parent grid. Select the desired data objects from the data explorer, and the data will be displayed in the 2D Viewer preview.

Enter a unique name for the child grid in the **Name** field.

Next, select the refinement ratio from the **Ratio** combo box. A ratio of 3:1, for example, will refine the parent cell by a factor of three, resulting in nine horizontal child cells within one parent cell.

Finally, specify the **Row Refinement** interval and the **Column Refinement** interval, by selecting the starting row/column and ending row/column, for where the grid refinement should be applied within the parent grid. The child grid can be placed anywhere within the parent grid as long as it does not overlap another child grid.

Note: The child grid cannot be rotated; it must be in the same orientation as the parent grid.

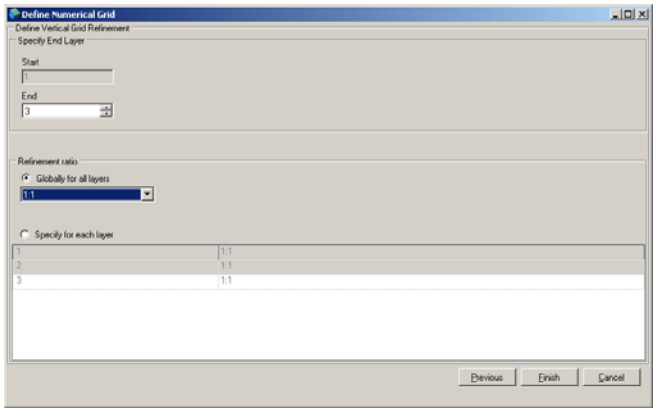
Click the **[Preview]** button to preview the child grid in the adjacent 2D Viewer.

Click the **[Next]** button to proceed to the next step.

Vertical Grid Refinement

Vertical grid refinement involves selecting which model layers to refine and specifying the refinement ratio for the selected layers

.



The top of the child grid must always coincide with the parent grid and therefore the **Start** layer will always be 1. However, the **End** layer can be any model layer below the top model layer in the simulation domain.

There are two options for defining the refinement ratio. Select **Globally for all layers** to assign a single refinement ratio to all layers. Alternatively, select **Specify each layer** to assign a refinement ratio layer by layer.

Note: Although the top layer must be the start layer, vertical refinement does not have to start at the top. Assign a refinement ratio of 1:1 to the top layer and it will not be refined.

Click the **[Finish]** button to create the child grid.

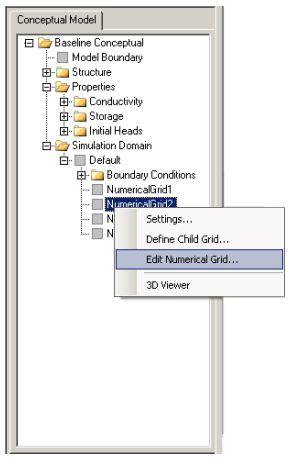
12.2 Editing a Finite Difference Grid

When a numerical grid is initially created (see “Creating a Finite Difference Grid” on page 175), the horizontal grid spacing is uniform. However, in many situations it is advantageous to have non-uniform grid spacing, to allow for finer grid discretization in the areas of interest, and larger grid spacing in areas which are less important, or where less data is available. Hydro GeoBuilder allows you to refine or coarsen areas of a numerical grid by adding or removing grid lines within a specified row/column interval. This process is described in the following section.

Note: You cannot refine/coarsen areas of a grid that overlap a child grid. It is recommended that you refine/coarsen the parent grid before creating a child grid.

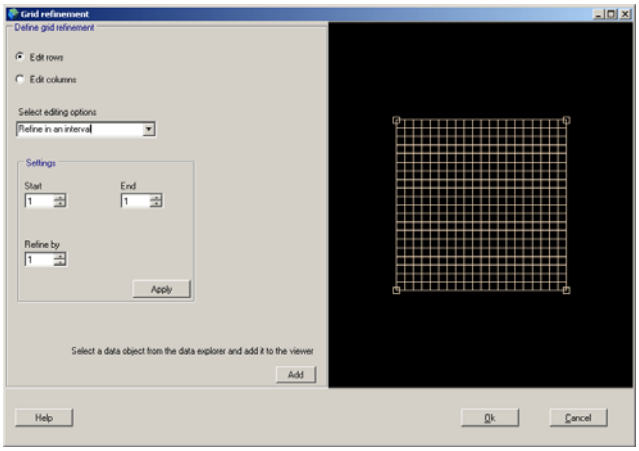
To edit the grid lines in a numerical grid, follow the steps below:

- From the **Conceptual Model Tree**, right-click on the **Numerical Grid**, and select **Edit** from the pop-up menu.



12.2.1 Horizontal Grid Refinement

The **Grid Refinement** dialog provides options for refining/coarsening a numerical grid. These options are described below.



Tip! You can add data objects (e.g., boundaries, wells, site maps) from the data explorer to the **2D Viewer** preview to assist you in determining the areas in which horizontal refinement/coarsening should be applied. Select the desired data objects from the **Data Explorer**, and the data will be displayed in the 2D Viewer preview.

Select the **Edit Rows** radio button to add/remove grid lines (rows) along the Y-direction of the numerical grid, or select the **Edit column** radio button to add/remove grid lines (columns) along the X-direction of the numerical grid.

From the **Select Editing Options** combo box, select either **Refine in an interval** or **Coarsen in an interval**. Refining in an interval will *add* rows/columns within a specified interval, where coarsening in an interval will *remove* rows/columns in a specified interval.

In the **Settings** frame, define the row/column interval for which the refinement/coarsening should be applied, by specifying the **Start** row/column and the **End** row/column. For example, if you would like to refine the grid area between row 20 and 30, you would enter 20 as the start row and 30 as the end row.

Finally, specify the refinement/coarsening factor in the **Refine by** box. For example, if *refine in an interval* is selected, a factor of 2 would subdivide each row/column within the specified interval into two equally spaced rows/columns. If *coarsen in an interval* is selected, a factor of 2 would reduce the number of rows/columns within the specified interval by a factor of 2.

Click the **[Apply]** button to show the defined refinement/coarsening in the adjacent 2D Viewer.

12.3 Deleting a Numerical Grid

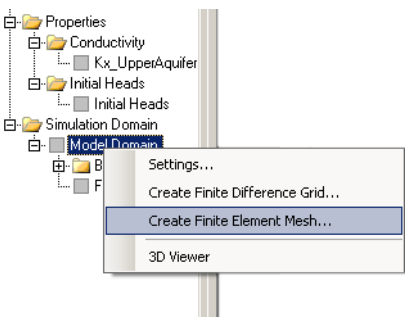
To delete a numerical grid, right-click on the grid in the **Conceptual Model** tree, and then select the **[Delete]** item from the pop-up menu.

12.4 Creating a Finite Element Mesh

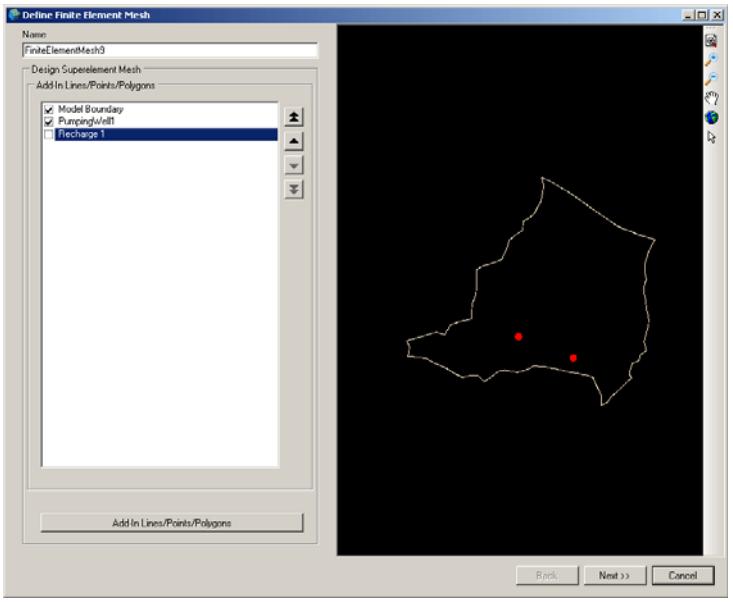
A finite element mesh is required for translating your conceptual model to a finite element numerical model, e.g., for loading into FEFLOW. The following section describes how to generate a finite element mesh, and the various options available.

To create a finite element mesh, follow the steps below:

- In the conceptual model tree, right-click on the **Model Domain** node, and select **Create Finite Element Mesh** from the pop-up menu.



The **Define Finite Element Mesh** wizard will appear on your screen, where you can configure the properties of the finite element mesh.



Specify a unique name for the finite element mesh in the **Name** text field.

Defining the Superelement Mesh

The **Superelement Mesh** represents the main geometry (points and segments) of the model region from which finite elements are generated. A superelement mesh is defined using the model boundary polygon geometry and, optionally, one or more "add-ins". Add-ins are lines, points or polygons within the model boundary which Hydro GeoBuilder uses as focal points to create nodes during finite element mesh creation.

By default, the **Add-ins List** contains the model boundary and any linear or point boundary condition currently defined for the conceptual model. Additional add-ins may be added to the list using data objects from the **Data Explorer**.

Note: To avoid unstable mesh designs, it is recommended that all add-in objects are pre-processed such that there is equal spacing between vertices on polygons and polylines. Data object geometry can be edited using the 2D Viewer editing tools. For more information, please see "Digitizing & Editing Geometry in 2D Viewers" on page 107.

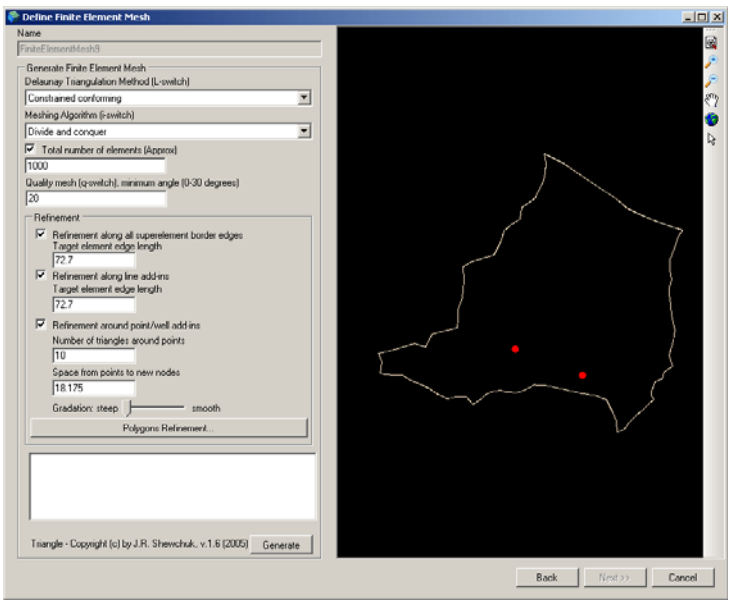
To add an add-in using a data object from the **Data Explorer**,

- Leaving the **Define Finite Element Mesh** window open, select the desired data object from the **Data Explorer**.
- In the **Define Finite Element Mesh** window, select the **Add-in Lines/Points/ Polygons** button, located below the **Add-ins List** .

An Add-in may be included or excluded in the mesh creation, by checking or unchecking the corresponding check box, respectively. When an add-in is “checked” it will also be displayed in the adjacent 2D Viewer preview window.

12.4.1 Defining the Horizontal Mesh Settings

In the second dialog in the **Define Finite Element Mesh** wizard, you can define various discretization settings for the horizontal mesh.



Finite element meshes are created using the Triangle mesh generator, developed by J.R. Shewchuk. Triangle provides various options for generating finite element meshes. These options are described briefly below.

Please note that the switch letters in parenthesis beside the name of each option refers to the command line switches used by the Triangle mesh generator.

Delaunay Triangulation Method (L-switch)

Delaunay triangulation methods are typically used in finite element mesh generation as they tend to maximize the minimum angle of all the angles of the triangles in the

triangulation, providing a nice set of triangles, and avoiding narrow “skinny” triangles. There are three triangulation methods from which to choose from (Table 1): **Constrained**, **Constrained Conforming** and **Conforming**.

Table 1: Delaunay Triangulation Methods (L-switch)

Method	Description
Constrained	Triangulation in which each segment appears as a single edge in the triangulation. As such, segments are not subdivided, and new vertices are not added to the vertex set. A constrained Delaunay triangulation is not truly a Delaunay triangulation, because some of its triangles might not be delaunay.
Constrained Conforming	Triangulation in which triangles are constrained delaunay, however, additional vertices may be added to the vertex set and segments may be subdivided to ensure a user-defined **Minimum Angle** constraint is satisfied. If a minimum angle is not specified, vertices are added to ensure all angles are between 20 and 140 degrees.
Conforming	Triangulation in which each triangle is truly delaunay, and not just constrained delaunay. Additional vertices may be added to the vertex set to enforce the delaunay property.

Meshing Algorithm (I-switch)

Two Delaunay algorithms are provided for generating the finite element mesh: **Divide and Conquer**, and **Incremental**. Typically, the divide and conquer algorithm is preferred. However, if this algorithm fails, use the incremental algorithm.

Total Number of Elements (Approx)

For the **Conforming** and **Constrained Conforming** triangulation methods, you can specify the desired number of elements that comprise the finite element mesh. Please note that the specified number of elements cannot be less than the default number generated by Triangle, *i.e.*, the number generated if this option is disabled.

Minimum Angle

For the **Constrained Conforming** triangulation method, a minimum angle can be specified. The specified angle will replace the default bound on the minimum angle (20 degrees). The specified angle may include a decimal point, but cannot be expressed in exponential notation.

Refinement Options

Edges of triangles along model boundary should have approx length

Use this option to set the approximate length of edges (segments) that comprise the model boundary domain. Vertices will be added along the boundary, creating subsegments with the specified length. This option will refine the areas along the model domain boundary.

Edges of triangles along line should have approx length

Use this option to set the approximate length of edges of triangles along line add-ins. Vertices will be added along the lines, creating subsegments with a specified length. The option will refine areas around line add-ins.

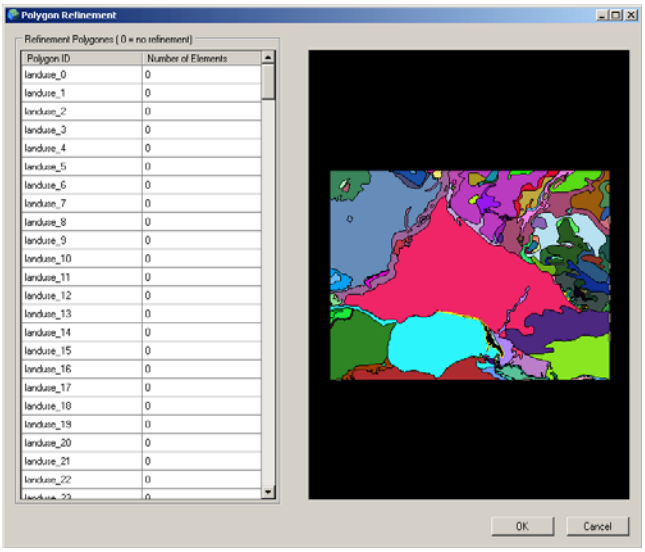
Refinement around point add-ins

Use this option to refine areas around point add-ins. Refinement for point add-ins is defined by specifying the number of triangles directly around the points, and the desired distance from the point to the new vertices.

Use the **Gradation** slider bar to specify the smoothness of the transition from the fine elements around the points to the coarser elements. A smoother transition will result in more elements, but will lead to more regular elements therefore improved model stability.

Polygon Refinement

You can use one or more polygon data objects that have been included in the superelement mesh (see “Defining the Superelement Mesh” on page 185) to define localized areas of mesh refinement. To do so, click the **Polygons Refinement...** button. The **Polygon Refinement** screen will appear.

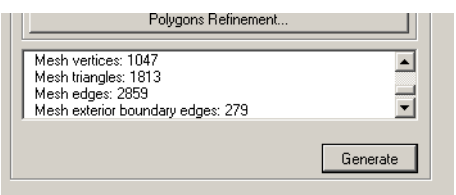


In the left side of the **Polygon Refinement** window, all features in the polygon add-ins are listed in a table under the **Polygon ID** column. When a feature is selected, it is highlighted yellow in the adjacent 2D Viewer. Select the polygon feature that represents that area of the mesh for which refinement is to be performed. In the adjacent field under the **Number of Elements** column, enter the desired number of element that should exist in polygon area. 3D-Builder will automatically refine the underlying triangles to equal the prescribed number of elements for the polygon area.

Once you have defined the refinement for the polygon features, click the **[OK]** button to apply the settings.

Generating the Horizontal Mesh

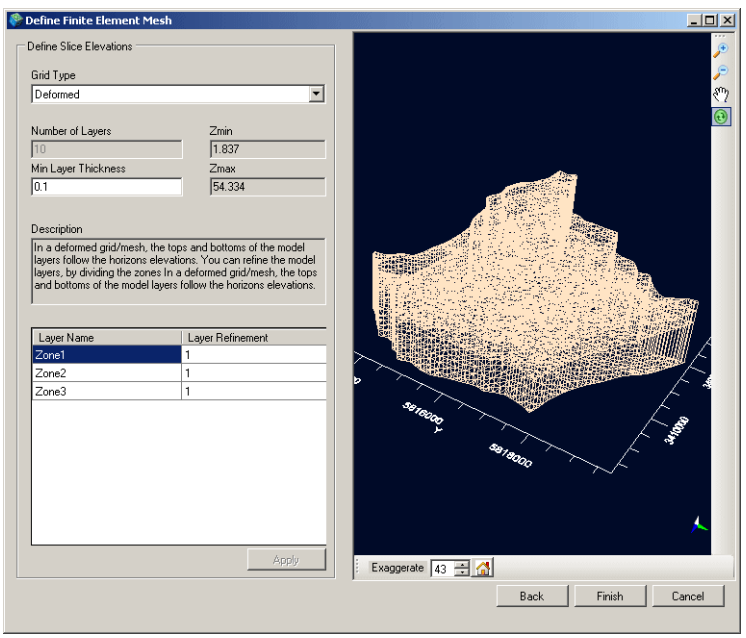
Once the above settings have been defined, click the **[Generate]** button to generate the horizontal mesh. Once generated, the Triangle output results, *e.g.*, number of mesh vertices, triangles, edges, etc, are displayed in the text box (above the Generate button). These results may be copied to the clipboard by right-clicking anywhere inside the box, and selecting **Copy to Clipboard** from the pop-up menu.



The generated mesh will also display in the adjacent 2D Viewer window. If you are not satisfied with the mesh, you can modify the settings and regenerate the mesh by selecting the **Generate** button again.

12.4.2 Defining Slice Elevations

The third step in the finite element mesh creation involves defining slice elevations. Various settings are available for defining the slice elevations in the third dialog of the finite element mesh creation wizard (shown below).



The options available for defining slice elevations are similar to those for defining slice elevations for finite difference grids. For information on these settings, *e.g.*, grid types, min layer thickness and layer refinement, please refer to “Defining the Vertical Grid” on page 177.

Once the vertical mesh settings have been defined, click the **[Finish]** button to generate the finite element mesh. The finite element mesh will now appear in the **Conceptual Model** tree under the **Model Domain** node, where it can be displayed in a **2D** or **3D Viewer** window, and used in finite element model translation.

13

Translating to Numerical Model

Hydro GeoBuilder allows you to translate your conceptual model to a finite difference model, for running in Visual MODFLOW, or to a finite element model for running in FEFLOW. During translation, Hydro GeoBuilder automatically populates the specified grid or mesh with the defined geological formations, boundary conditions and property zone attributes, and creates the necessary input files for loading into your desired simulator. For finite difference models, Hydro GeoBuilder will generate the MODFLOW input data files required for loading into Visual MODFLOW. For finite element models, Hydro GeoBuilder will generate the .FEM problem file for loading into FEFLOW.

This chapter walks you through the steps involved in translating a conceptual model to a numerical model, and includes information on the following topics:

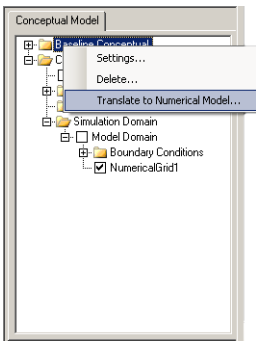
- Translating a Conceptual Model to a Numerical Model
- Defining Simulation Settings
- Choosing Boundary Condition Packages
- Importing into Visual MODFLOW
- Importing Initial Heads from a MODFLOW Model
- Limitations of MODFLOW Data Import
- Running MODFLOW-LGR Simulation

13.1 Translating a Conceptual Model to a Numerical Model

13.1.1 Translating to MODFLOW

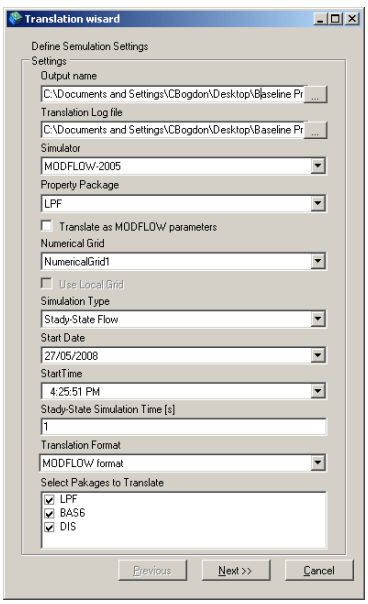
To translate your conceptual model to a numerical model, follow the steps below.

- From the **Conceptual Model** tree, right-click on the desired conceptual model, and select **Translate to Numerical Model** from the pop-up menu.



- The translation wizard will launch, where you can define the simulation settings. These settings are described in the following section.

Defining Simulation Settings



Output Name

Click the ... button and specify the output name and directory for the *.NAM file that is created during translation.

The .NAM file is an ASCII file containing a list of the input and output data files for the model, and their location (folder and path name) on the computer. This file is required for importing your model into Visual MODFLOW.

The default file name is “Project_name.NAM” and the default directory on your computer is [Project Folder]\Numerical Models.

Translation Log File

When a numerical model is translated in Hydro GeoBuilder, a log file is automatically generated and saved on your computer. By default the log file name is [Project_Name].LOG and the directory is [Project Name]\Numerical Models.

Click the ... button to specify a new file name and directory.

Simulator

Currently Hydro GeoBuilder supports translation for the following numeric engines:

- MODFLOW-2000
- MODFLOW-2005
- MODFLOW-2005 Local Grid Refinement (LGR). See *“Running MODFLOW-LGR Simulation” on page 203*

Property Package

Select the type of property package to use for translation. Choose either the Layer Property Flow (**LPF)** package or the Block Center Flow (**BCF6)** package.

Numerical Grid

Select which numerical grid to use from the combo box. The combo box contains all numerical grids created for the conceptual model.

Simulation Type

Select **Steady State** or **Transient** from the combo box.

If the **Steady State** option is selected, Hydro GeoBuilder will prepare the data set for a steady-state flow simulation, and will automatically use the data from the first stress period of each boundary condition and pumping well defined in your conceptual model.

If the **Transient Flow** option is selected, Hydro GeoBuilder will automatically merge all the different time period data defined for each pumping well and boundary condition into the stress period format required by the different versions of MODFLOW.

Start Date

The default start date is the date specified in the conceptual model settings. The Start Date of the model is the date corresponding to the beginning of the simulation. currently, this date is relevant only for transient flow simulations where recorded field data may be used for defining time schedules for selected boundary conditions.

Start Time

Specify the simulation start time. The Start Time of the model is the time corresponding to the beginning of the simulation. Currently, this time is relevant only for transient flow simulations where recorded data may be imported for defining time schedules for selected boundary conditions.

Steady-State Simulation Time

A steady state simulation time is required for Steady State Flow simulation. This parameter is not used if you have selected Transient Flow. Although the simulation will always be run to the same equilibrium solution in Steady State, the total amount of water passing through boundary conditions (i.e. the cumulative value of the solution) depends on the amount of time simulated.

Translation Format

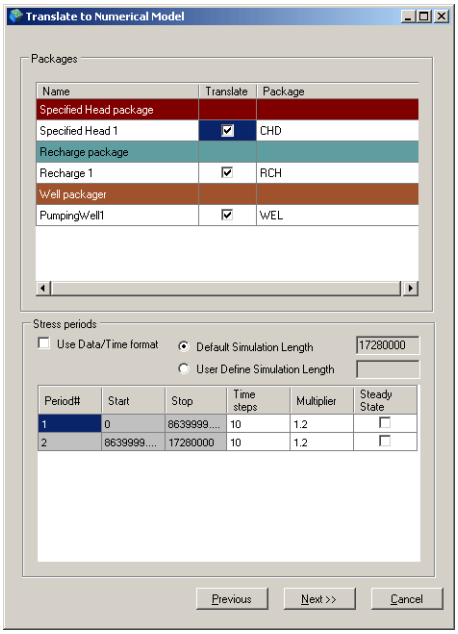
Currently, only the MODFLOW translation is supported for the finite-difference calculations.

Select Packages to Translate

Select which packages to translate by checking or unchecking the desired check boxes. For example if you only want to translate the discretization package, select just the **DIS** package.

Click the **[Next]** button to proceed to the next step.

Choosing Boundary Condition Packages



The **Packages** frame lists the existing boundary conditions for the conceptual model being translated. Here you can include/exclude boundary conditions by selecting the appropriate check box in the **Translate** column.

If at least one boundary condition contains transient data, and the simulation type was set to transient in the previous step, the options in the top of the **Stress Periods** frame will be active.

Select the **Use Date/Time** format to express the **Start** and **Stop** time using absolute date and time format.

Choose to use either the default simulation length, calculated by Hydro GeoBuilder, or define a new simulation length by selecting the **User Defined Simulation Length** radio button, and by entering a new length.

The grid at the bottom of the **Stress Periods** frame contains the stress periods for the simulation. If the simulation is transient, Hydro GeoBuilder automatically calculates the stress periods from the defined time schedule data in the boundary conditions. If a boundary condition is excluded or included in the **Packages** frame, the stress periods will be recalculated accordingly. If the simulation is steady state, there will only be one stress period.

The **Time Steps** column is used to define the number of time steps in each stress period. By default each stress period is divided into 10 steps. The multiplier column is used to increment each time step size, *i.e.*, it is the ratio of the value of each time step to that of

the preceding time step. The default value is 1.2. A time step Multiplier value greater than 1 will produce smaller time steps at the beginning of a stress period resulting in a better representation of the changes of the transient flow field. Thus increasing the number of time steps in a simulation may result in smoother head or drawdown versus time curves.

The **Steady-State** column indicates if the stress period is transient or steady-state. MODFLOW allows individual stress periods in a single simulation to be either transient or steady state instead of requiring the entire simulation to be either steady state or transient. Steady-state and transient stress periods can occur in any order. Commonly the first stress period may be run as steady state, to produce a solution that is used as the initial condition for subsequent transient stress periods.

Click the **[Next]** button to translate the model to the numerical model.

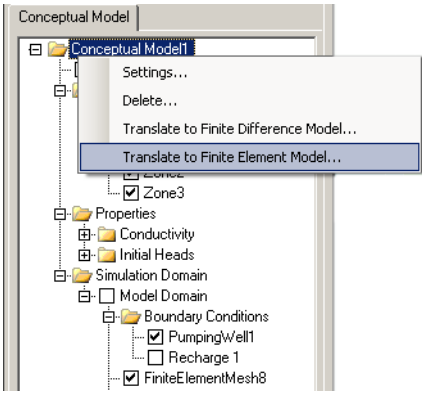
Translate to Numerical Model
Translation Log Details
========== Translation Start ==========
DIS Package ...
#Discretization Package translator: Initialize
#Discretization Package translator - (c) Schlumberger Water Services
Translate plan geometry
Translate Top Elevation
Translate Bottom Elevation for Layer# 1
Translate Bottom Elevation for Layer# 2
Translate Bottom Elevation for Layer# 3
Translate StressPeriods
#Discretization Package translator: Finalize
LPF Package ...
#Layer Property Flow Package translator: Initialize
#Layer Property Flow Package translator - (c) Schlumberger Water Services
Translate Property for Layer #1
Translate Property for Layer #2
Translate Property for Layer #3
Previous
Next >>
Cancel

While the model is being translated, the log details are displayed in the dialog box (shown above). Once the translation is finished, click the **[Next]** button to close the dialog box.

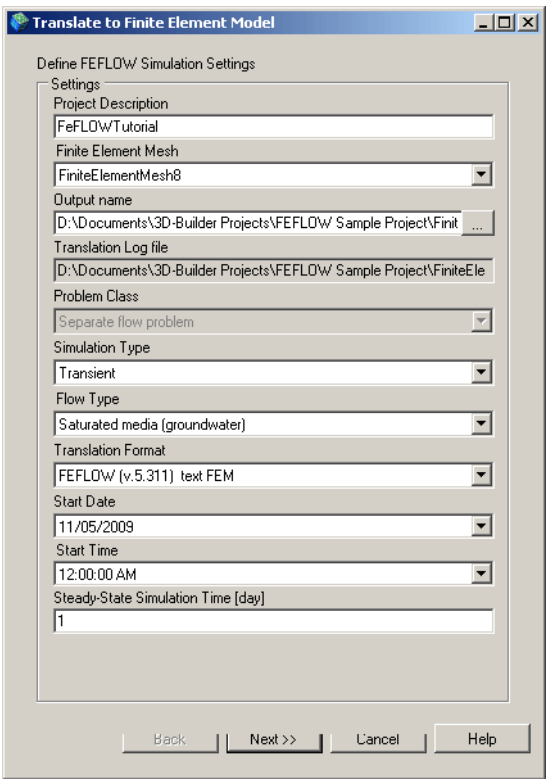
13.1.2 Translating to FEFLOW

To translate a conceptual model to a finite element numerical model,

- In the **Conceptual Model Tree**, right-click on the desired conceptual model folder, and select **Translate to Finite Element Model...** from the pop-up menu..



Define FEFLOW Simulation Settings



Project Description

A brief description of the project. By default, this is the name of the current Hydro GeoBuilder project.

Finite Element Mesh

Select the desired finite element mesh from the combo box to use for model translation.

Output Name

Click the ... button and specify the output name and directory for the *.FEM file that is created during translation.

The .FEM file is an ASCII file containing information on the problem class and model properties. This file is required for importing your model into FEFLOW.

The default file name is “Project_name.FEM” and the default directory on your computer is [Project Folder]\Numerical Models.

Translation Log File

When a numerical model is translated in Hydro GeoBuilder, a log file is automatically generated and saved on your computer. By default the log file name is [Project_Name].LOG and the directory is [Project Name]\Numerical Models.

Click the ... button to specify a new file name and directory.

Problem Class

Currently, 3D-Builder only supports the separate flow problem class.

Simulation Type

Select **Steady State** or **Transient** from the combo box.

If the **Steady State** option is selected, 3D-Builder will prepare the data set for a steady-state flow simulation, and will automatically use the data from the first stress period of each boundary condition and pumping well defined in your conceptual model.

If the **Transient Flow** option is selected, 3D-Builder will automatically merge all the different time period data defined for each pumping well and boundary condition into the stress period format required by FEFLOW.

Flow Type

Select the flow type of the problem class. Choose from the following options: **Saturated media (groundwater)**, **unsaturated media**, **Unsaturated steady-state linearized Richards equation**.

Translation Format

The output .FEM file generated by 3D-Builder during translation is currently compatible with FEFLOW v.5.3.11 or later.

Start Date

The default start date is the date specified in the conceptual model settings. The Start Date of the model is the date corresponding to the beginning of the simulation. This date is relevant only for transient flow simulations where recorded field data may be used for defining time schedules for selected boundary conditions.

Start Time

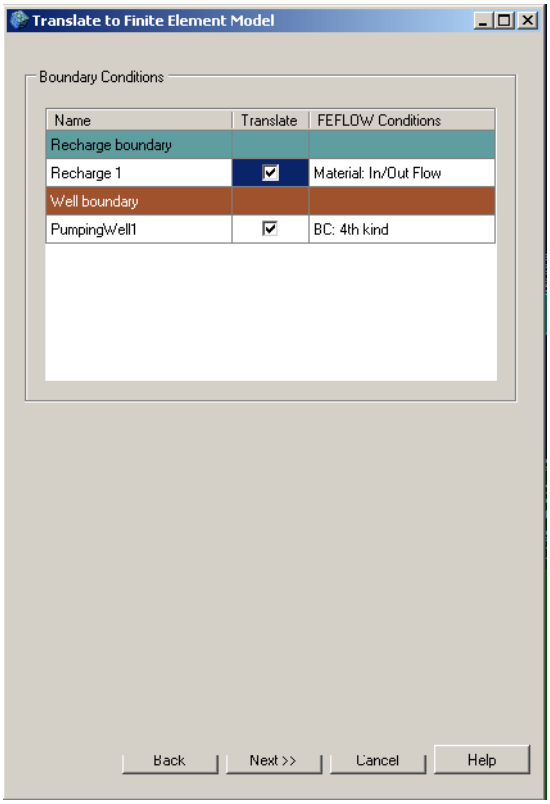
Specify the simulation start time. The Start Time of the model is the time corresponding to the beginning of the simulation. This time is relevant only for transient flow simulations where recorded data may be imported for defining time schedules for selected boundary conditions.

Steady-State Simulation Time

A steady state simulation time is required for Steady State Flow simulation. This parameter is not used if you have selected Transient Flow. Although the simulation will always be run to the same equilibrium solution in Steady State, the total amount of water passing through boundary conditions (i.e. the cumulative value of the solution) depends

on the amount of time simulated.

Packages



The **Packages** frame contains a list of existing boundary conditions for the conceptual model being translated. Here you can include/exclude boundary conditions by selecting the appropriate check box in the **Translate** column. The corresponding FEFLOW condition type is listed for each boundary condition under the FEFLOW Conditions column.

Click the **[Next]** button to initiate the model translation. During translation, output details are displayed in the translation log. Once finished, click the **[Finish]** button to exit the translation wizard.

13.2 Importing into Visual MODFLOW

If you have translated the model using MODFLOW-2000 or MODFLOW-2005, you can import the model into Visual MODFLOW for running the simulation. The import routine is described in section Importing MODFLOW Data Sets in the Visual MODFLOW User's Manual.

Importing Initial Heads from a MODFLOW Model

When Visual MODFLOW imports a MODFLOW data set, the Initial Heads array is not automatically read into the Initial Heads property array. Instead, an .HDS file is created using the file name format [projectname].VMP.HDS. You may specify this Initial Head file from the Run Menu, under **MODFLOW-Initial Heads**. Please see "Initial Heads" section in the Visual MODFLOW User's Manual for more information on using the **Previous Visual MODFLOW Run** option.

Limitations of MODFLOW Data Import

When importing MODFLOW models into Visual MODFLOW, some settings may not be retained. For a summary of the limitations of MODFLOW data import, please see the section "Limitations of MODFLOW data import" in the Visual MODFLOW User's Manual.

Note: When specifying the data set units (Start Date, Start Time, Conductivity, Pumping Rate, Recharge), be sure that the selected units match those specified in the Hydro GeoBuilder.

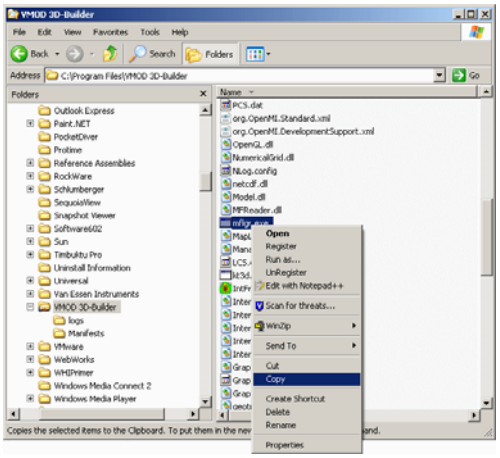
13.3 Running MODFLOW-LGR Simulation

Currently, Visual MODFLOW does not support MODFLOW-LGR simulations. However, you can run the simulation using the USGS MODFLOW-LGR executable. This executable is copied to your computer during the Hydro GeoBuilder installation to the Hydro GeoBuilder installation directory.

To run a MODFLOW-LGR simulation, follow the steps below.

[1] Right-click on the Window's start button, and select Explore to open a new Window's Explorer window.

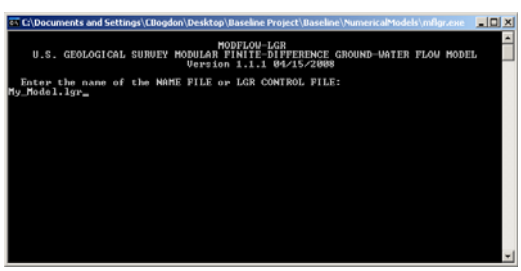
[2] Navigate to the Hydro GeoBuilder Program Files folder, e.g., c:\Program Files\VMOD 3D-Builder, locate the file **mflgr.exe,** right-click on it and select **Copy.**



[3] Now open the folder that contains the translated MODFLOW-LGR files. By default this location should be [Data Repository]\Numerical Models. Select **Edit** from the main menu, and click **Paste**.

[4] Now that the mflgr.exe file is located in the same folder as the translated MODFLOW-LGR files, you can run the simulation. To do so, double-click on the **mflgr.exe** file to a launch the executable.

[5] A DOS prompt window will display on your screen. Type in the full name of the translated LGR file, *e.g.*, [ProjectName].LGR, and then press the **Enter** key on your keyboard.


C:\Documents and Settings\CBogdon\Desktop\Baseline Project\Baseline\NumericalModels\mflgr.exe
MODFLOW-LGR
U.S. GEOLOGICAL SURVEY MODULAR FINITE-DIFFERENCE GROUND-WATER FLOW MODEL
Version 1.1.1 04/15/2008
Enter the name of the NAME FILE or LGR CONTROL FILE:
My_Model.lgr

[6] During the simulation, the output files, *i.e.*, LST and HDS, are created and saved for both the parent grid and each child grid, in the same location as the translated input files (where the **mflgr.exe** was copied to). Once the simulation is finished, you can open the **LST** file in a text editor, *e.g.*, **Notepad**, for each grid to view details of the simulation including the mass balance. Additionally, you can import the HDS file into Hydro GeoBuilder as a 3D-Gridded data object, to visualize the simulated heads in a 3D-Viewer window. For more information on importing

HDS files, please refer to “Importing 3D Gridded Data” on page 41.

Note: If the LST files were not generated after running the simulation, it could be because the folder path containing the specified LGR file is too long, e.g., C:\Documents and Settings\User\Desktop\Modeling Project\Groundwater Model\NumericalModels. If this is the case, it is reccomended that you copy and paste the contents of the folder into a folder that has a shorter path, e.g., C:\Temp, and then rerun the simulation.

13.1 Viewing Results in Hydro GeoBuilder

Viewing Finite Difference Model Output

Once you have translated and run your model using Visual MODFLOW, you can import some of the output results back into Hydro GeoBuilder for visualization.

Importing Heads File

After running a simulation in Visual MODFLOW, the heads information is stored in the **projectname.HDS** file. This file can be imported into your Hydro GeoBuilder project using the Import 3D Gridded Data process (see “Importing 3D Gridded Data” on page 41). Once imported, you can interpolate heads data onto a surface data object using the create new attribute operation (see “Creating an Attribute from 3D Gridded Data Object” on page 72). After applying this operation, you can then color and display contours on the surface data object using the surface style settings (see “Surfaces” on page 88)

Importing Pathlines

If you added particle tracking in your simulation (using Visual MODFLOW), you can export the pathlines to a shapefile (in Visual MODFLOW), and then import the shapefile into Hydro GeoBuilder using the Import Polyline import process (see “Importing Polylines” on page 30).

For information on how to export GIS data from Visual MODFLOW, please refer to the Visual MODFLOW User’s Manual.

Importing Head Contours

Head contours can be viewed in Hydro GeoBuilder by first exporting the output head contours to a shapefile (in Visual MODFLOW), and then importing the shapefile into Hydro GeoBuilder using the Import Polyline import process (see “Importing Polylines” on page 30).

For information on how to export GIS data from Visual MODFLOW, please refer to the Visual MODFLOW User's Manual.

Appendix A: Supported Data Types

Data Type	Supported File Types	Description	How can it be used in Hydro GeoBuilder?
Points	.XLS, .MDB, .DXF, .TXT, .CSV, .ASC	Discrete data points with known attribute(s), *e.g.*, X, Y, elevation, top/bottoms of formations, Kx, Initial Heads.	Interpolate the points to generate surfaces, which can be used for defining conceptual model horizons, or distributed parameter values such as Kx, Initial Heads, Recharge, etc.
Polygons	2D/3D ESRI Shapefile, AutoCAD DXF	GIS vector files containing polygon geometry and attributes	Use to define the conceptual model domain Use to delineate property zones Use to define geometry of aerial boundary conditions, *e.g.*, lake, recharge, specified-head.
Polylines	2D/3D ESRI Shapefile, AutoCAD DXF	GIS vector files containing line geometry and attributes	Use to define geometry of linear boundary conditions, *e.g.*, river, drain, general head.
Surfaces	USGS .DEM, ESRI ASCII Grid (.ASC, .FRD), Surfer .GRD (ASCII or Binary)	Files containing an ordered array of interpolated values at regularly spaced intervals that represent the spatial distribution of an attribute, *e.g.*, digital elevation model	Use to define conceptual model horizons Use to assign spatially-variable attributes to boundary conditions and property zones.
Wells	.XLS	Well head coordinates (X, Y, Z) and associated well attribute data such as screen intervals, pumping schedules, observation points and data, well tops (contact points with geological formations), and well path (for deviated wells)	Interpolate well heads to generate surface representing topography. Convert well tops to surfaces representing top/bottoms of geological formations Use to define pumping well boundary conditions.

Data Type	Supported File Types	Description	How can it be used in Hydro GeoBuilder?
Time Schedules	.XLS	Attributes measured over time, *e.g.*, hydrographs	Use to define transient data for boundary conditions, such as recharge, river stage elevations etc.
Maps	.JPG, .BMP, .TIF, .GIF	Raster images, *e.g.*, aerial photographs, topographic maps, satellite imagery	Use sitemaps for gaining a perspective of the dimensions of the model, and for locating important characteristics of the model.
Cross Sections	HGA-3D Explorer (.3XS)	Cross sections generated using Hydro GeoAnalyst data management software	Generate surfaces from cross section model interpretation layers and use for defining model horizons/structural zones.
3D Gridded Data	TecPlot . DAT, MODFLOW .HDS	3D Grid with attributes at each grid cell.	Use to visualize heads data generated from a MODFLOW run in Visual MODFLOW. Use to assign spatially-variable attributes to boundary conditions and property zones.

Index

Numerics

3D-Builder
  Features 2
  Workflow 7

A

Add Data Object 177
Add Points 107
Add Polygon 107
Add Polyline 107
Add-ins 185
Artificial Withdrawal 153

B

Background Color 100
Boundary Condition Location 158
Boundary Conditions
  Define 155
  Drain 149
  Evapotranspiration 151
  General Head 146
  Lake 152
  Pumping Well 142
  Recharge 151
  River 144
  Specified Flux 154
  Specified Head 143
By Attribute 78

C

Cascade 98
Child Grid 180
  Column Refinement 181
  Refinement Ratio 181
  Row Refinement 181
Color By Attribute 82
Conceptual Model
  Advantages 1
  Create 118
  Delete 120
  General 117
Conceptual Model Settings 119
Conceptual Model Tree 119
Connection Elevation 160
Constrained 187
Contour Lines 90
Control Points 49
Create New Data Object 106
Creating a new project 15
Cutaways 101

D

Data Import
  Data Mapping 27
  Data Validation 29
  General 23
  Multiplier 28
Data Objects
  Delete 56
  Export 56
Data Operations 70
  Arithmetic 71
  Attribute 72
  Cross Sections 74
  Wells 74
Data Repository
  General 16
Data Settings
  General 62
Data Table 63
Data Viewers
  Export 105
  General 95
  Layer Ordering 100
  Open 95
Define Pumping Well Boundary Condition 155
Defining Boundary Condition Parameters 165
Deformed Grid 178
Deformed-Uniform Grid 179
Distance to Reservoir 148
Divide and Conquer 187
Drains
  Conductance 149
  Drain Elevation 149

E

Edit Geometry 107
Equal Intervals 83, 89
Evaporation Rate per Unit Area 153
Evapotranspiration 151
Evapotranspiration rate 152
Export
  Data Objects 56
Extinction Depth 152

F

Facets 164
Finite Element Mesh 184

G

General Head 147
  Conductance 147
General Head Average Conductivity 148
Georeferencing 48
  Options 50
Global 164
Graticule 52

H
Hardware Requirements 5
Horizon Types
  Base 121
  Conformable 122
  Discontinuity 121
  Erosional 121
Horizons
  General 121

I
Import 3D-Gridded Data 41
  Grid Origin 43
  Grid Rotation 43
Import Cross Sections 44
  General 44
Import Maps 46
Import Polygons 32
Import Polylines 30
Import Surfaces 33
Import Time Schedules 53
  Absolute 54
  Relative 54
Import VMOD data 57
Import Wells 34
  Observation Points 38
  Pumping Schedule 37
  Screens 37
  Well Heads Only 36
  Well Tops 39
Import wells
  Well Path 40
Incremental 187
Installation 5
Interpolation Mode 52
Interpolation Settings 112
Interpretation Type 84
Inverse Distance 112

J
JPEG Image Quality 51

K
Kriging 113

L
Label Placement 81
Lakebed Conductivity 153
Lakebed Thickness 153
Layer Refinement 178
Leakance 145
Licencing 7

M
Minimum Angle 187
Minimum Cell Thickness 178
MODFLOW-LGR
  General 180
Modify Project Settings 18

N
Natural Neighbors 113
Number of Classes 83, 89
Numerical Grid
  Coarsening 183
  Create 175
  Delete 184
  Edit 182
  Grid Rotation 176
  Refinement 183
  Types 178

O
Observation Points 68
Online Help 12
Open a Project 21
OpenGL Driver 103
Overland Runoff 153

P
Pick Mode 104
Point Style 103
Polyline Zones 161
Precipitation Rate per Unit Area 153
Project Coordinates 17
Project Description 17
Project Name 16
Property Parameters 20
Property Zone Parameters
  Constant Value 134
  Use 3D Gridded Data 134
  Use Shapefile 135
  Use Surface 134
Property Zones
  Assign Parameters 132
Propery Zones
  Create New 130
  General 129
Pumping Well Schedule 67

R
Rescale 107
River Stage 145
River Width 145
Riverbed Bottom 145
Riverbed Kz 145
Rivers
  Conductance 145
  River Bottom Elevation 145
Rotate 107

S
Save a Project 21
Show Area Fill 79

Show Labels 81
Spreadsheet 63
Stage 147
Start Date 118
Statistics 63
Structural Zones 127
Styles Settings 76
Superelement Mesh 185
Surfaces
  Create 110
  General 109
  General Settings 110

T

Tile 98
Triangulation
  Delaunay 186

U

Uniform Grid 178
Uninstalling 6
Units 18, 19
  Conversion 28
Use a polygon extent 114
User Interface 10
  Main Window 10

V

Vertical Exaggeration 101
View Mode 104
Virutal Grid 103

W

Well Head 66
Well Path 69
Well Screens 66
Well Table 65
  Horizontal Wells 69
  Vertical Wells 65
Well Tops 68